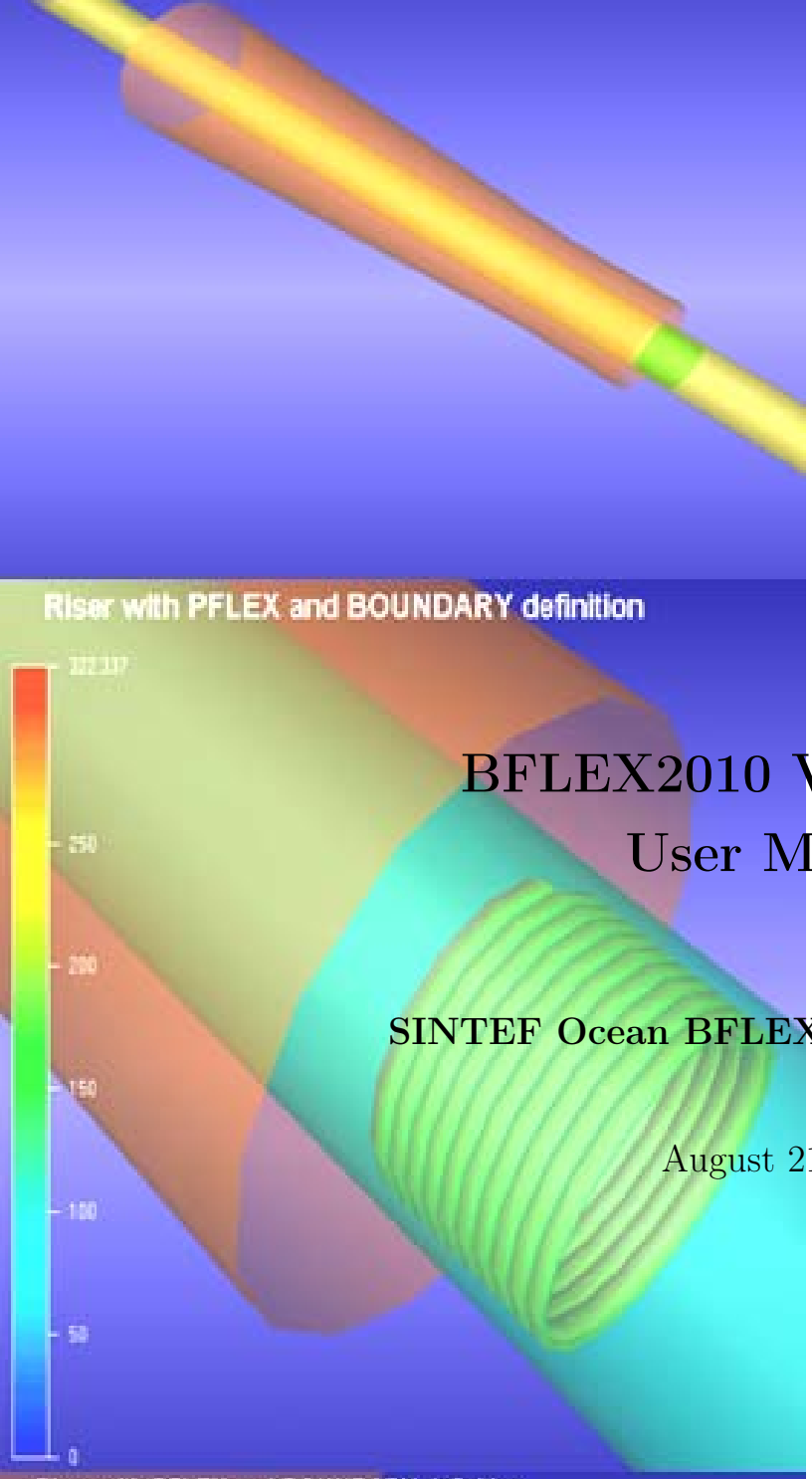
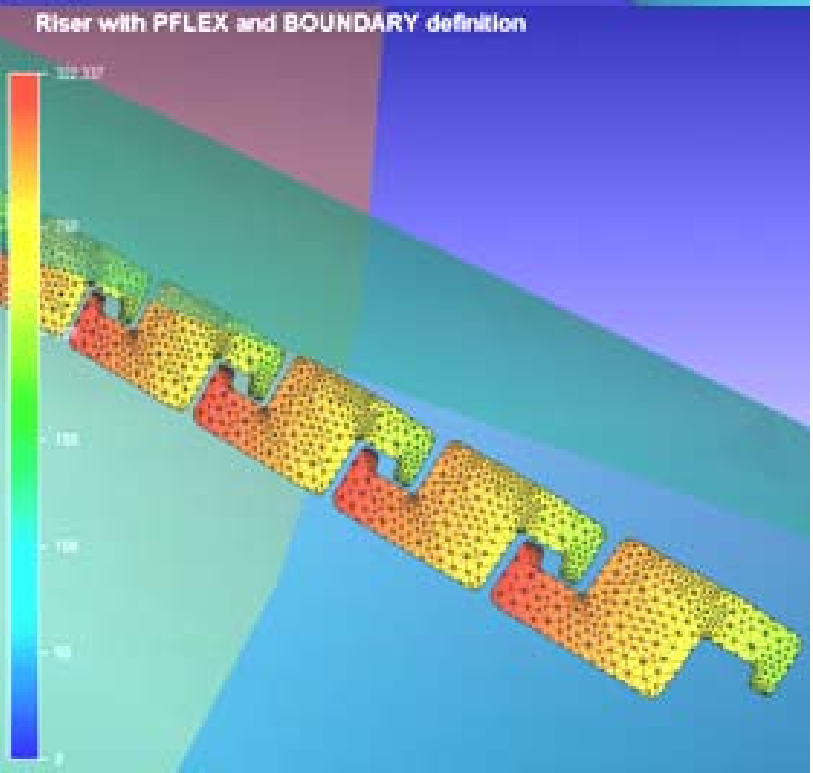




BFLEX2010 Version 3.3.1
User Manual

SINTEF Ocean BFLEX Development Team

August 21, 2024

Contents

Contents 1

List of Figures 7

List of Tables 9

1 Introduction 10
1.1 Theoretical background . . . 10
1.2 Purpose of document . . . 11
1.3 General . . . 11
1.4 Program documentation . . . 12
1.5 Program structure . . . 13
1.6 BFLEX2010 program files . . . 13
1.7 How to run BFLEX2010 . . . 14
1.8 Backgrounds related to cross-section stress analysis in BFLEX . . . 16

2 BFLEX2010 input guide 22
2.1 General . . . 22
2.2 BONCON - Boundary Conditions . . . 27
2.3 CLOAD - Concentrated loads . . . 28
2.4 CONSTR - Constraints . . . 29
2.4.1 PDISP . . . 29
LOCAL . . . 30
GLOBAL . . . 30
2.4.2 CONEQ . . . 31
LOCAL . . . 31
LOCSL . . . 32
GLOBAL . . . 32
2.4.3 HISTCONEQ . . . 33
2.5 CONTINT - contact interfaces . . . 34
2.6 CONTROL - CONTROL parameters . . . 37
2.6.1 STRESSFREE . . . 38
2.6.2 RESTART . . . 38
2.6.3 EIGEN . . . 38
2.6.4 AUTOSTART . . . 39

2.7 COSUPR Contact surface material properties . . . 41
2.8 COSURFPR - Contact surface properties . . . 42
2.8.1 Standard route file . . . 43
2.9 CROSSGEOM - Cross sectional geometry . . . 44
2.9.1 PIPE . . . 45
2.9.2 BOX . . . 45
2.9.3 GENERAL . . . 46
2.9.4 BFLEX . . . 48
2.10 CROSSECTION- BFLEX2010 Cross sectional input . . . 52
2.10.1 FLEXCROSS . . . 53
2.10.2 353FLEXCROSS . . . 57
2.10.3 BENDSTIFF . . . 61
2.10.4 BENDSTIFF-BEND . . . 61
2.10.5 NLBENDSTIFF . . . 62
2.10.6 NLBENDSTIFF-BEND . . . 62
2.10.7 Fatigue Data . . . 63
2.11 DYNCONT - Control parameters for dynamic analysis . . . 67
2.12 DYNRES_ - dynamic results . . . 68
2.12.1 DYNRES_N . . . 68
2.12.2 DYNRES_E . . . 69
2.12.3 DYNRES_I . . . 70
2.13 ELCON - Element connectivity and properties . . . 71
2.13.1 PIPE . . . 74
2.13.2 COMPIPE . . . 75
2.13.3 CABLE . . . 75
2.13.4 CONT . . . 75
2.13.5 SPRING . . . 76
2.13.6 HELIX . . . 76
2.13.7 HSHEAR . . . 77
2.13.8 HCONT . . . 78
2.13.9 HELSPR . . . 78
2.13.10 SEA . . . 78
2.13.11 BODY . . . 79
2.14 ELDAMP - Element damping properties . . . 79
2.14.1 BEAM . . . 80
2.14.2 CONTACT . . . 80
2.14.3 RALEIGH . . . 81
2.15 ELECC - Element eccentricity . . . 82
2.15.1 BEAM . . . 82
2.15.2 STINGER . . . 83

2.15.3 RADIUS . . . 85
2.16 ELHIST - element time histories . . . 88
2.17 ELLOAD - Element loads . . . 90
2.18 ELMASS - Element mass properties . . . 91
2.19 ELORIENT - Orientation of elements . . . 92
2.20 ELPROP - Element properties . . . 95
2.20.1 PIPE . . . 96
2.20.2 COMPIPE . . . 98
2.20.3 BELLMOUTH . . . 99
2.20.4 BODY . . . 100
2.20.5 CABLE . . . 101
2.20.6 ROLLER . . . 102
2.20.7 GENSPRING . . . 105
2.20.8 HELSPRING . . . 106
2.20.9 HELIX . . . 107
2.20.10 SZHELIX . . . 107
2.20.11 LAYERCONTACT . . . 109
2.20.12 SHEARHELIX . . . 110
2.20.13 SHEAR2HELIX . . . 111
2.20.14 SHEARMODEL . . . 112
2.20.15 SOILCONTACT . . . 115
2.21 ENVRES_ - Envelope results . . . 117
2.21.1 ENVRES_N . . . 117
2.21.2 ENVRES_E . . . 118
2.21.3 ENVRES_I . . . 118
2.22 FATPROP - FATigue properties . . . 119
2.22.1 Fatigue file format . . . 120
2.23 GEOM - geometry of body . . . 121
2.24 HEAD - HEADing . . . 123
2.25 INISTR - initial strain loading . . . 123
2.26 MATERIAL - MATERIAL properties . . . 124
2.26.1 LINEAR (PIPE31, HELIX231, 234, FLEXCROSS, 353FLECROSS, PIPE31, HSHEAR353, 363, 364) . . . 126
2.26.2 ELASTIC (FLEXCROSS, 353FLEXCROSS, HSHEAR353, 363, 364) . . . 127
2.26.3 ELASTOPLASTIC (FLEXCROSS, PIPE33, HELIX233) . . . 127
2.26.4 EPCURVE . . . 129
2.26.5 HYCURVE . . . 131
2.26.6 ISOHYCURVE . . . 132
2.26.7 SEA . . . 133

2.26.8 RESULTANT (COMPIPE42, HELIX235) . . . 133
2.26.9 CONTACT (CONT126, 130, 152, HELIX231, 233, 234, 235)) . . 135
2.26.10 ISOCONTACT (CONT130, CONT152, CONT164) . . . 139
2.26.11 ISOKXYCONTACT (CONT164) . . . 140
2.26.12 FRICONTACT (HCONT453, 454, 463, 464 and 473) . . . 140
2.26.13 FRICVISC (HCONT463 and 464) . . . 142
2.26.14 R_CONTACT (CONT126) . . . 143
2.26.15 HYPERELASTIC (FLEXCROSS) . . . 148
2.26.16 GENSPRING (SPRING137) . . . 149
2.26.17 HELSPRING (HELSPR437) . . . 150
2.27 MOVE_GROUP - move group during autostart . . . 150
2.28 NOCOOR - Nodal coordinates of model . . . 150
2.29 NODPROP - node interpolated element properties . . . 153
2.30 NOORIENT - orientation of nodes . . . 155
2.31 PELOAD - external pressure and gravity loading . . . 156
2.32 PILOAD - internal pressure load . . . 157
2.33 REEL - reeling and straightening simulation scenario definition . . . 159
2.34 TABLE - Table data . . . 161
2.35 THIST - time histories . . . 163
2.35.1 THIST_F - history from file . . . 163
2.35.2 THIST_H - harmonic load . . . 163
2.35.3 THIST_R - ramping loads smoothly on or off . . . 164
2.36 TIMECO - time control data . . . 165
2.37 TLOAD - temperature loading . . . 166
2.38 VISRES - visual results . . . 167

3 BFLEX2010POST Report Generator 170
3.1 General . . . 170
3.2 Input data . . . 171
3.3 NOPLOT - NOdal history PLOTs . . . 171
3.4 IPPLOT - Integration Point history PLOTs . . . 172
3.5 ELPLOT - ELement history PLOTs . . . 173
3.6 GNPLOT - Global Nodal PLOTs . . . 176
3.7 GLPLOT - Global ELement PLOTs . . . 177
3.8 FAPLOT- Fatigue calculation . . . 180
3.9 NRPLOT- Nodal Reaction PLOTs . . . 180
3.10 RSPLOT - ReStart PLOT . . . 182
3.11 ENPLOT- ENvelope PLOTs . . . 183
3.12 BFPOST - BFlex POSTprocessing . . . 183
3.13 GRPLOT - ELement Reaction PLOTs . . . 184

4 DYNPOSTB Report Generator 186
4.1 General . . . 186
4.2 Input data to DYNPOSTB . . . 187
4.2.1 MXPLOT - MaXima/MiNima postprocessing . . . 187
4.2.2 DYNPLOT - Dynamic plot postprocessing . . . 188

5 PFLEX Input Guide 189

6 BOUNDARY input guide 192

7 LIFETIME input guide 194
7.1 Format of the life time data (.lif file) . . . 194

8 BPOST Report Generator 196
8.1 General . . . 196
8.2 Input data . . . 196
8.3 NOPLOT - NOdal history PLOTs . . . 196
8.4 NOPLO2 - NOdal history PLOts 2 . . . 198
8.5 ELPLOT - ELement history PLOTs . . . 200
8.6 GNPLOT - Global Nodal PLOTs . . . 201
8.7 GLPLOT - Global ELement PLOTs . . . 203

9 Examples 205
9.1 General remarks . . . 205
9.2 2D cross-section stress analysis of a 4 inch flexible pipes . . . 205
9.2.1 Objectives . . . 205
9.2.2 Input data . . . 206
9.2.3 Modelling . . . 207
9.2.4 Results in terms of axisymmetric stresses . . . 208
9.2.5 Results in terms of bending moment and friction stresses . . . 209
9.3 Stress behaviour of flexible riser at the platform hang-off . . . 213
9.3.1 Objectives . . . 213
9.3.2 Input data . . . 213
9.3.3 Modelling . . . 214
9.3.4 Results . . . 215
9.4 Lateral buckling behaviour of tensile armours . . . 216
9.4.1 Objectives . . . 216
9.4.2 Input data . . . 217
9.4.3 Modelling . . . 217
9.4.4 Results . . . 218

9.5 Lateral buckling behaviour of tensile armours taking lateral contact into account . . . 219
9.5.1 Objectives . . . 219
9.5.2 Input data . . . 220
9.5.3 Modelling . . . 220
9.5.4 Results . . . 222

10 Searching for Errors 227

11 Changelists 228
11.1 Changes in version 3.3.0 . . . 228
11.2 Changes in version 3.2.0 . . . 229
11.3 Changes in version 3.1.1 . . . 231
11.4 Changes in version 3.1.0 . . . 232
11.5 Changes in version 3.0.9 . . . 236

Bibliography 238

Index 240

List of Figures

1.1 BFLEX2010 system architecture. 15

2.1 Coordinate system. 24
2.2 Coordinate system. 25
2.3 Definition of integration points for tubes 25
2.4 Definition of integration points for rectangles 26
2.5 Definition of corners 1-4, and integration points (P1,P2) in the old raf file format 26
2.6 Position of pipe in free span analysis (**ICATEN = 0**). 41
2.7 Vessel definition. 41
2.8 Input format for a route file with 3 lines. 44
2.9 Illustration of line definition for a route file with 3 lines. 44
2.10 **CROSSGEOM** of type **PIPE**. 46
2.11 **CROSSGEOM** of type **BOX**. 46
2.12 **CROSSGEOM** of type **GENERAL**. 47
2.13 **CROSSGEOM** *pipegen* of type **GENERAL** from example. 48
2.14 Axis systems. 50
2.15 Segment description. 50
2.16 **CROSSGEO** of type **BFLEX**. 51
2.17 Cross section integration point (IP) definition 70
2.18 Element Gaussian station numbers s1 to s3 along the element axis. Integration points P1 and P2. 71
2.19 Element coordinates (x, y, z), roller coordinates (x_1, y_1, z_1), orientation angle φ of the roller coordinate system and incremental change of arch length ds and orientation angle dφ between rollers. 85
2.20 Modelling of spiral with linearly changing radius of curvature. 86
2.21 Rotation-symmetric geometry for pipe-pipe external contact interaction modelled by the **RADIUS** option. 88
2.22 Rotation-symmetric geometry modelled by the **RADIUS** option. . . 89
2.23 **ELORIENT** command for PIPE and HSHEAR elements. 93
2.24 **ELORIENT** command for the HELSPR437, CONT, BODY and SPRING elements. 94
2.25 Defining maximum allowable incremental pipe contact point displacement by **CONTPAR2=VALUE** 104

2.26 Analysis where **CONTPAR1=1** and **CONTPAR2=0** or **1** is recommended . . . 104
2.27 Helix coordinate system. . . . 108
2.28 Szhelix coordinate system. . . . 109
2.29 Illustration of material hardening effect. . . . 128
2.30 Material curve combination. . . . 139
2.31 Polar coordinate definition. . . . 153
2.32 **NOORIENT** command. . . . 156
2.33 Reeling. . . . 160

9.1 Comparing σ_{zz} with respect to different contact algorithm at internal pressure 20MPa . . . 209
9.2 Comparing σ_{xx} with respect to different contact algorithm at internal pressure 20MPa . . . 209
9.3 Comparing moment curvature obtained by alternative modelling procedures against test data . . . 210
9.4 Zoom of moment curvature obtained by alternative modelling procedures versus test data . . . 211
9.5 The corresponding axial friction wire stress at outer fibre in inner armour during one curvature cycle . . . 211
9.6 Zoom of the corresponding axial friction wire stress at outer fibre in inner armour during one curvature cycle . . . 212
9.7 Model applied to simulate tensile armour stresses at bend stiffener section of flexible riser . . . 213
9.8 Stress distribution at end fitting for alternative helix FE models . . . 215
9.9 Comparison of curvature and stress distributions using alternative helix FE models . . . 216
9.10 Model for tensile armour lateral buckling . . . 218
9.11 Components of stress . . . 219
9.12 Development of torsion and stress for a tensile armour buckling - failure load case . . . 220
9.13 Development of torsion and stress for a tensile armour buckling - no failure load case . . . 220
9.14 The tensile armour buckling process . . . 221
9.15 RUC Model for tensile armour lateral buckling . . . 222
9.16 Development of stress during tensile armour buckling . . . 223
9.17 Stress and gap distributions around circumference . . . 223

List of Tables

2.1 Overview of BFLEX2010 element library. 73
2.2 Units for **EPS** and **SIGMA**. 129

9.1 CROSS-SECTION PARAMETERS - 4 INCH RISER ID = 101.6 MM 206
9.2 MATERIAL PROPERTIES - 4 INCH RISER 206
9.3 ALTERNATIVE MODELLING PROCEDURES - 4 INCH RISER . 208
9.4 INPUT PARAMETERS - SIMULATION OF FLEXIBLE RISER . . 214
9.5 FLEXIBLE PIPE DATA 224
9.6 TEST CONDITIONS 225
9.7 NUMERICAL RESULTS VERSUS TEST RESULTS IN TERMS OF FAILURE/NO FAILURE 225
9.8 FLEXIBLE PIPE DATA 226

1 Introduction

The BFLEX computer program and its submodules was originally developed by SINTEF Civil and Environmental Engineering as part of the project Service Life Analysis of Deepwater Risers which was sponsored by the Testrig JIP (oil companies, 70%) and The Norwegian Research Council (30%).

The first program version was released in 2001. A BFLEX User Group was further established in order to commercialize and maintain the program system. The user group first included Statoil, Norsk Hydro and Seaflex. Since then, several new users have been included and the user group of today additionally includes NOV, Statoil, Kongsberg Maritime, ExxonMobil, Aker Solutions, 4Subsea, Shanghai Jiaotong University (SJTU), 2H Offshore, FlexLife and Prysmian.

In a user group meeting arranged 2007-11-29 a new development was initiated in order to:

- Provide better modelling flexibility (e.g. enabling modelling gaps between bend stiffener and pipe).
- Ensure that the BFLEX Program System is able to provide industry needs beyond 2010.
- Improve BFLEX with respect to feedback from existing users.

The new computer program system is termed BFLEX2010.

In 2023 the USAP software was merged with BFLEX2010. USAP was developed for Aker Solutions, with tailormade helix-elements to analyse umbilical cross sections with long pitch lengths and limited direct interaction between the individual components.

1.1 Theoretical background

For details regarding the theoretical background, see the theory-manuals, (Sævik, 2022) and (Sævik, 2 14b). Extensive testing has been varried out to verify the program, see e.g. (Sævik et al., 1998), (Sævik and Igland, 2002), (Sævik, 2011), (Sævik and Li, 2013), (Dai et al., 2017), (Dai et al., 2020) and (Sævik, 2 14a).

1.2 Purpose of document

The purpose of the present report is to describe the input required in order to perform analysis with BFLEX2010 version 3.3.1 and to post-process the results.

1.3 General

BFLEX2010, BFLEX2010POST, DYNPOSTB, PFLEX, BOUNDARY, BPOST and LIFETIME are tailormade program modules for stress and fatigue analysis of flexible pipes. BFLEX2010 performs the global and tensile armour analysis. PFLEX performs beam stress analysis of the pressure spirals, BOUNDARY performs transverse stress analysis of the pressure spirals and LIFETIME performs fatigue calculations. In addition results postprocessing and 3D colour plots are enabled by the postprocessors BFLEX2010POST, DYNPOSTB, BPOST and the graphical user interface XPOST. BFLEX2010POST also provides the link between the new modelling features represented by BFLEX2010 and the original BFLEX functionality from 2001 including the PFLEX, BOUNDARY, BPOST and LIFETIME.

BFLEX2010 and the post processors BFLEX2010POST and DYNPOSTB offer fairly general capabilities with regard to modelling and post processing of helical slender structures, as compared to the original BFLEX version which only considered one section of a flexible pipe. In BFLEX2010 the original BFLEX functionality with regard to flexible pipe modelling part is included by means of the **FLEXCROSS** feature, see Section 2.10.1 in combination with element types **PIPE52** and **HSHEAR352**. BFLEX2010POST can further be applied to generate a result file on the old BFLEX raf file format for parts of the model. This can further be used in stress and fatigue analysis using PFLEX (Sævik, 1999), BOUNDARY (Sævik et al., 2001) and BPOST. The PFLEX and BOUNDARY models only include a limited number of pitches representing a single user selected location along the global BFLEX2010 tensile armour model(submodelling approach), applying the global response from BFLEX2010 as boundary conditions. Having obtained the stress variation from these analyses, LIFETIME performs fatigue analyses based on the obtained stress ranges and given fatigue data for each layer and for each nodal point in the local model generated from BFLEX2010POST. BPOST can perform postprocessing of the local BFLEX2010 results in terms of *e.g* ovalization and the gap results stored in the aforementioned local raf file.

If no evaluation of the pressure armour is needed, the stress time series for the helices can be processed directly by using the **DYNRES** functionality see Section 2.12 in com-

bination with DYNPOSTB. This enables full resolution stress time series to be generated at user selected locations, without significant impact on computation speed.

A summary of the available modules is given below:

- BFLEX2010 module, reading and controlling all input data needed for all modules, and performing global as well as tensile armour stress analysis.
- DYNPOSTB module, for postprocessing of user selected time series stored on the binary dyn database to ASCII files, see Section 2.12.
- BFLEX2010POST module, for results postprocessing from the binary raf database to ASCII files that can be plotted by the plotting program such as Matrixplot. BFLEX2010POST also offers the convertion of user selected sections of the BFLEX2010 model into the old (raf-file) format for further processing applying:
  - PFLEX module, performing pressure spiral bending stress analysis.
  - BOUNDARY module, performing transverse cross-sectional stress analysis.
  - LIFETIME module, performing fatigue analysis.
  - BPOST module, for results postprocessing from the local results database on ASCII files that can be plotted by the plotting program Matrixplot.

The XPOST graphical user interface for result presentation can be applied to view results from both the BFLEX2010 and the local model generated by BFLEX2010POST.
It should be noted that for the local raf-file, a text string "_bflex2010" is added to the prefix in order to highlight that this is a local postprocessed BFLEX model.

1.4 Program documentation

The program documentation consists of the following reports:

- BFLEX2010 program system, User's manual (this document)
- BFLEX2010POST program system, User's manual (this document)
- BFLEX2010 Theory Manual, (Sævik, 2022).
- USAP Theory manual, (Sævik, 2 14b).
- PFLEX and BOUNDARY Theory Manual, (Sævik, 2022).
- PFLEX, BOUNDARY and LIFETIME User's Manual (this document)
- BPOST program system, User's manual (this document)

1.5 Program structure

A schematic overview of the system architecture for the BFLEX2010 program system is shown in Fig. 1.1.

The main parts of the BFLEX2010 program system are:

BFLEX2010 BFLEX2010 analysis module

BFLEX2010POST BFLEX2010POST postprocessing module (binary raf database)

DYNPOSTB DYNPOSTB time series postprocessing module (binary dyn database)

PFLEX PFLEX analysis module

BOUNDARY BOUNDARY analysis module

LIFETIME LIFETIME analysis module

BPOST The local model postprocessing module

XPOST Graphical user interface for result presentation

1.6 BFLEX2010 program files

The input and output files name convention:

prefixname.2bif BFLEX2010 Input File (model definition)

prefixname.bof BFLEX2010 Output File (print of result from input data handling plus results)

prefixname.blf BFLEX2010 Log File (print of the iteration status at each load step) and error messages.

prefixname.2bpi BFLEX2010POST Input File

prefixname.2bpl BFLEX2010POST Log File

prefixname.bdi DYNPOSTB Input File

prefixname.bdo DYNPOSTB Log File

prefixname.plf PFLEX Log File (print of the iteration status at each load step)

prefixname.bol BOUNDARY Log file (print of the iteration status at each load step)

prefixname.raf BFLEX2010 Result Database File (result presentation)

prefixname_bflex2010.raf BFLEX2010 Result Database File generated by BFLEX2010POST for selected parts of the global model. It can be postprocessed by PFLEX, BOUNDARY, LIFETIME and BPOST.

prefixname.mpf BFLEX2010 Matrix Plot File (optional)

fatiguedata BFLEX2010 fatigue data file which is arbitrarily named by the user (one for each layer type where fatigue analysis is to be carried out)

prefixname.lif LIFETIME Input File

prefixname.lof LIFETIME Output File

prefixname.bpi BPOST Input File

prefixname.bpl BPOST Log File

prefixname.pif PFLEX Input File

prefixname.boi BOUNDARY Input File

Files with extensions such as geo, msh, bdf and dat are geometry and mesh definition of profiled components such as armor wires.

1.7 How to run BFLEX2010

BFLEX2010 can either be executed through command line or FLEXEDIT on Windows machines.

A description on how to setup and run BFLEX2010 from FLEXEDIT is available in the BFLEX2010 *Quick Start User Guide*.

When running BFLEX2010 from the command line on Windows machines, the basis is to write the name referring to the BFLEX2010.EXE in the prompt. In addition either of the following options can be applied:

- Adding -s and a float number (larger than 0.001) multiplies the default maximum number (5000) of load steps allowd by BFLEX2010. Increasing the step numbers will result in longer simulation time and larger result file size.
- Adding -m and a float number (> 0.001) multiplies the default memory used by BFLEX2010 by the factor given.
- Adding -n and the input file prefix name you want to run makes BFLEX2010 run with this file name without asking for it in the prompt.

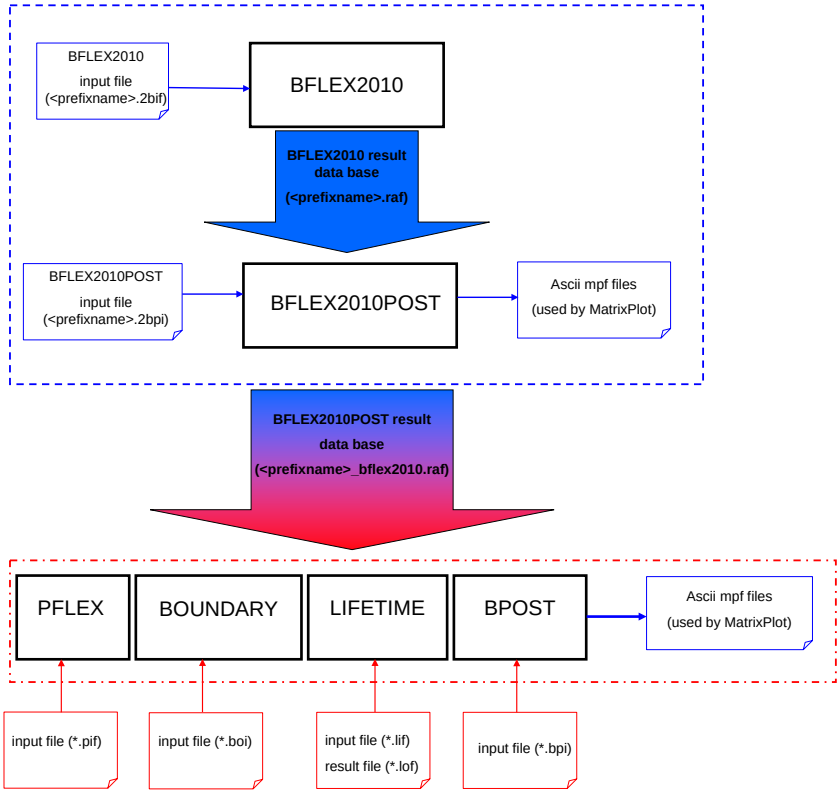


Figure 1.1: BFLEX2010 system architecture.

- Adding -d and a float number (> 0.001) representing the multiplication factor for scaling of the default memory size for **DYNRES** results used by BFLEX2010.
- Adding -s2 and an integer number representing the number of steps required for dynres storage, only relevant when this result type is exported by the **DYNRES** card. Default value is 15000.

In an analysis restart, BFLEX2010 must be executed with the same -m, -s and -d multiplication factors as applied in the previous analysis, and also the same -s2 integer number.

EXAMPLE:

```
command line: BFLEX2010 -s 2 -m 1.56 -n prefixname
```

After a successsfull run of BFLEX2010, results from the binary *.raf file can be viewed using XPOST or extracted to ASCII format using BFLEX2010POST, ref. Section 3.

BFLEX2010POST can also be used for creating a new *_bflex2010.raf database that can

be further postprocessed by PFLEX, BOUNDARY, LIFETIME and BPOST ref. Section 5, Section 6, Section 7 and Section 8.

Note: BOUNDARY must be executed before PFLEX

1.8 Backgrounds related to cross-section stress analysis in BFLEX

BFLEX was origininally developed to enable lifetime evaluations of North Sea flexible risers due to armour metal fatigue. This effort was based on the PhD work by (Sævik, 1992) and supported by the oil industry through several industry and joint industry projects until now. The latest developments finalized in 2019 were related to enabling lateral buckling calculations of deep water risers.

Most models used to predict stresses due to bending of helical structures are based on the assumption that no 3D end effects are present so that a 2D assumption can be applied for each cross-section. This means that the behaviour at a given cross-section is only governed by the kinematics and constitutive relation given at that cross-section alone which is a reasonable assumption whenever the curvature gradient is small and the curved region is sufficiently far away from the end fitting constraint where each wire is terminated. BFLEX2010 supports both the 2D assumption, by application of cross-section stress resultant models (MCM models) and the 3D assumption, by application of sandwich beam based models (SBM models) considering the forces and moments in each individual wire. It is to be noted that even in the latter case, no ovalisation effects are included in the BFLEX tensile armour models.

The original BFLEX modelling approach included both the moment-curvature (MCM) based models, see Eq. (2.2), known to industry as **ITCODE21** and **ITCODE31**, and the individual helix sandwich beam model approach (SBM), known to industry as **ITCODE0** and **ITCODE1**, see Eq. (2.3) and Section 2.20.14. These models are linked to the **PIPE52** beam element and the **HSHEAR352** sandwhich beam element. In both cases, the axisymmetric stresses and layer contact pressures are found by applying the CAFLEX model, see (Feret and Momplot, 1989) which was developed as a cooperation between SINTEF and IFP as part of the FPS2000 project, see (Berge et al., 1992). The CAFLEX model assumes concentric layers and Lagrange multipliers to resolve layer contact pressures. Many years later, similar 2D models have been published in the litterature, see e.g. (Skeie et al., 2012) resulting in the Helica software. The slip curvature and displacement necessary to address the friction effect for the MCM and SBM models, respectively, are calculated at the load step corresponding to the mean static load. Local elastic stresses are calculated by analytical theory as a user defined linear interpolation between the geodesic and loxodromic curve assumptions. This is also the case for the **ITCODE0** and **ITCODE1** SBM models since only axial slip is allowed

for. The **ITCODE0** SBM model was developed first and calibrated against outer tensile armour strain gauge data (Sævik et al., 1998), then followed by the **ITCODE21** MCM model which was calibrated against fibre optic bragg strain data for the inner tensile armour, see (Sævik and Igland, 2002). For both cases and to save computing time, the stick slip friction characteristics is based on the inner armour contact pressure and friction properties. Later on, following the developments in computer capabilities, this has been improved to include one stick slip model for each armour layer resulting in the **ITCODE31** MCM and **ITCODE1** SBM models, leading to improved accuracy in fatigue prediction, see (Sævik, 2011).

Then in order to enable more flexible modelling features, BFLEX2010 was developed, where the link to the BFLEX sub-modelling procedure for evaluating stress and fatigue in the pressure armour (via PFLEX, BOUNDARY, LIFETIME) is taken care of by the **FLEXCROSS** cross-section modelling feature, see Section 2.10.1, allowing for all of the above **ITCODE** options.

The way the moment-curvature and shear stress-slip displacement curves is established (fixed slip curvature/displacement) for the **ITCODE** approaches can lead to numerical problems due to undesired stiffness variations. This is normally not a concern for shallow water risers as the North Sea ones, however, numerical instabilities have been experienced for deep water risers with large tension variations associated to each curvature cycle. Therefore an improved friction model based on constant stick stiffess has been included as an option for all models, see (Dai et al., 2018). The moment-curvature stick bending stiffness can also be significantly affected by shear interaction which was described in (Sævik and Li, 2013). This can be taken into account by introducing user defined shear interaction parameters, see Section 2.20.14. However, this is only recommended to be applied in combination with the new constant stick stiffness friction model, as the original **ITCODE** procedures and associated friction models already include some shear interaction as they were tuned with respect to experimental strain data.

Later on, a completely new framework has been developed to support lateral buckling calculations and to resolve the uncertainties introduced by analytical curve assumptions related to wire local bending. This has been obtained by developing the **HSHEAR353**, **HSHEAR363**, **HSHEAR364**, **HCONT464**, **HCONT453**, **HCONT454**, , **HCONT463** and **HCONT473** finite elements, which allows for bi-directional slip and lateral contact as well as through thickness stress and contact effects to be addressed. To model a flexible pipe cross-section, the **353FLEXCROSS** cross-section modelling feature was introduced, see Section 2.10.2. However, this modelling concept does not include the link to the pressure armour sub-modelling procedure.

BFLEX2010 now offers a variety of 2D assumption models for calculating stresses due

to axisymmetric loads and bending in the tensile armour. Such models are easy to establish and offers high computation efficiency as only one or two beam segments along the length is needed, then applying the tension and symmetric end rotations (constant curvature) as boundary conditions at the segment ends. They are applicable both for establishing material curves for global analysis and for post-processing the global response quantities into time series of stress prior to rainflow counting and fatigue calculations. As of now, the following 2D analysis alternatives exist:

- The MCM based **ITCODE21** model referred to above. Both old (default) and new friction models are available. Linked to the **FLEXCROSS** automatic flexible pipe modelling feature, see Section 2.10.1.
- The MCM based **ITCODE31** model referred to above. Both old (default) and new friction models are available. Linked to the **FLEXCROSS** automatic flexible pipe modelling feature.
- The MCM based **ITCODE32** model which is similar to the above but without any shear interaction tuning, i.e. pure plane surfaces remain plane by analytical theory and no shear interaction effect. Both old (default) and new friction models are available. Linked to the **FLEXCROSS** automatic flexible pipe modelling feature.
- Manual penalty and MCM based modelling of a concentric layered structure by combining **HSHEAR364** and **HCONT464** elements. Only the new friction model is available. No link to the **FLEXCROSS** or **353FLEXCROSS** automatic flexible pipe modelling features exist, so manual modelling is required. Local bending stress in helix version of **HSHEAR364** is based on the loxodromic curve assumption only. Manual modelling requires specifying the element properties of all finite elements by applying the **SHEARHELIX**, **SHEAR2HELIX**, **LAYERCONTACT** and **CONTINT**, see Section 2.20.12, Section 2.20.13, Section 2.20.11 and Section 2.5. Then the material properties for the helices include the **ELASTIC** and **LINEAR** options wherease the contact interfaces need to be specified by the **FRICONTACT** material, see Section 2.26.1, Section 2.26.2, Section 2.26.12 and Section 2.26.13. It is noted that in order to get the correct axial and bending stiffness of circular armours, this has to be implemented by applying the linear material model, see Section 2.26.1. This is because the geometry description by **SHEAR2HELIX** includes only two options: TUBE (thin-walled assumption) and RECTANGLE (assumed filled), see Section 2.20.13.
- Repated Unit Cell (RUC) modelling, see (Lukassen et al., 2019) based on **ITCODE0** as referred to above. Both old (default) or new friction models are available.Linked to the **FLEXCROSS** automatic flexible pipe modelling.
- Repated Unit Cell (RUC) modelling based on **ITCODE1** as referred to above.

Both old (default) and new friction models are available. Linked to the **FLEXCROSS** automatic flexible pipe modelling.

- Repated Unit Cell (RUC) modelling based on combining **HSHEAR364** and **HCONT464** elements for cylindric layers and pressure spirals/tapes. Then combining **HSHEAR353** with **HCONT463** or **HCONT453** to establish SBM model for tensile armour layers. Both old (default) and new friction model are available. Linked to the **353FLEXCROSS** automatic flexible pipe modelling. For lateral buckling calculations also include **HCONT454** to describe contact within helix layers. No need for any slip assumption because bi-directional slip is allowed for.

- Repated Unit Cell (RUC) modelling based on applying the **HSHEAR363** element (the thin shell version of **HSHEAR364** without radial stresses) for cylindric layers and pressure spirals/tapes. Then combining **HSHEAR353** with **HCONT463** or **HCONT453** for tensile armour layers. Both old (default) and new friction models are available. Linked to the **353FLEXCROSS** automatic flexible pipe modelling. For lateral buckling calculations also include **HCONT454** to describe contact within helix layers. No need for any slip assumption because bi-directional slip is allowed for.

- Repated Unit Cell (RUC) manual modelling of other types of helical structures based on combining **HSHEAR364** element for cylindric layers and pressure spirals/tapes. Then combining **HSHEAR353** with **HCONT463** or **HCONT453** for helix layers. Several friction models are available. If sideways contact are deemed important, also include **HCONT454** to describe contact within helix layers. No need for any slip assumption because bi-directional slip is allowed for. This functionality has been applied for power cables, see (Boru, 2021). It is noted that in order to get the correct axial and bending stiffness of circular armours, this has to be implemented by applying the linear material model, see Section 2.26.1 as noted above. This is because the geometry description includes only two options: TUBE (thin-walled assumption) and RECTANGLE (assumed filled). Manual modelling requires the element properties of all finite elements to be specified by applying **SHEARHELIX**, **SHEAR2HELIX**, **LAYERCONTACT** and **CONTINT**, see Section 2.20.12, Section 2.20.13, Section 2.20.11 and Section 2.5 together with the relevant material properties.

The MCM models can also be used to model a few meters of the flexible riser including the bend stiffener to address the stiffness contributions from the tensile armour layers, then with inputs from global analysis in terms of tension and end angles.

In cases where the end fitting is close to bending gradients, the transient introduced by terminating each helix need to be considered. This requires each helix to be modelled

individually and a few meters to be modelled including the bend stiffener, applying tension and end angles from global analyses as input. BFLEX2010 today offers different 3D assumption models for addressing this issue:

- The SBM based **ITCODE0** model referred to above. Both old (default) and new friction models are available. Linked to the **FLEXCROSS** automatic flexible pipe modelling feature.

- The SBM based **ITCODE1** model referred to above. Both old (default) and new friction models are available. Linked to the **FLEXCROSS** automatic flexible pipe modelling feature.

- Model based on combining **HSHEAR364** and **HCONT464** elements and cylindric layers and pressure spirals/tapes. Then combining **HSHEAR353** with **HCONT463** or **HCONT453** to establish an SBM model for tensile armour layers. Both old (default) and new friction models are available. Linked to the **353FLEXCROSS** automatic flexible pipe modelling. For lateral buckling calculations also include **HCONT454** to describe contact within helix layers. No need for any slip assumption because bi-directional slip is allowed for.

- Model based on applying **HSHEAR363** for cylindric layers and pressure spirals/-tapes. Then combining **HSHEAR353** with **HCONT463** or **HCONT453** to establish SBM model for tensile armour layers. Both old (default) and new friction models are available. Linked to the **353FLEXCROSS** automatic flexible pipe modelling. For lateral buckling calculations also include **HCONT454** to describe contact within helix layers. No need for any slip assumption because bi-directional slip is allowed for.

- Manual modelling of other types of helical structures based on combining **HSHEAR364** or **HSHEAR363** with **HCONT464** for cylindric layers and pressure spirals/tapes. Then combining **HSHEAR353** with **HCONT463** or **HCONT453** for helix layers. Several friction models are available. If sideways contact are deemed important, also include **HCONT454** to describe contact within helix layers. No need for any slip assumption because bi-directional slip is allowed for. This functionality has been applied for 3D modelling of umbilicals, see (Dai et al., 2020). It is noted that in order to get the correct axial and bending stiffness of circular armours, this has to be implemented by applying the linear material model, see Section 2.26.1. This is because the geometry description in **SHEAR2HELIX** includes only two options: TUBE (thin-walled assumption) and RECTANGLE (assumed filled). Manual modelling requires specifying the element properties of all finite elements by applying the **SHEARHELIX**, **SHEAR2HELIX** and **LAYERCONTACT** in addition to the desired material properties, see Section 2.20.12, Section

2.20.13, Section 2.20.11, Section 2.26.2, Section 2.26.1 and Section 2.26.12.

The latest development includes the contact element **HCONT473** that consider contact between **HSHEAR353** and **HSHEAR363** or **HSHEAR364**, however, also including their respective centreline axial and torsion motions in the relative displacement and friction. The element allows for arbitrary sliding. This means that sliding of one helix relative to the remaining cross-section due to temperature, curvature and gravity can be assessed in addition to evaluating local bending stresses at the end fitting. Also the significant hysteresis observed for torsion motion and its coupling to the bending motion can be described.

2 BFLEX2010 input guide

2.1 General

All input data is described in ASCII file format. Comment lines are defined by a "#" in the first column, i.e. at the start of the line. Empty lines are ignored. The input is organized in different data groups with individual identifiers, as listed below. Data belonging to one identifier will be read until next identifier or end of file is reached, i.e. input can be broken over lines. The data groups can be supplied in arbitary order. The maximum line length is 136 characters and the maximum number of letters in one single text string is 32.

The input may be given by an arbitrary set of units if not otherwise noted. The user must ensure that a consistent unit set is applied.

During a restart analysis, some input data such as boundary conditions and loading may be changed by the user. Data identifiers that can be changed during restart are marked with *ITALIC* font. The following identifiers define the different data groups:

BONCON	Boundary conditions
CLOAD	Concentrated loads
CONSTR	Constraints between nodal DOFs
CONTINT	Contact interfaces
CONTROL	Control parameters
COSUPR	Contact surface properties (soil description)
COSURFPR	Contact surface properties (route file)
CROSSGEOM	Cross sectional geometry
CROSSECTION	BFLEX2010 cross sectional input
DYNCONT	Control parameters for dynamic analysis
DYNRES_	Dynamic result presentation
ELCON	Element connectivity and properties
ELDAMP [(1)]	Element damping properties
ELECC	Element eccentricities
ELLOAD	Element loads
ELMASS	Element mass properties
ELORIENT	Orientation of elements
ELPROP	Element properties

ENVRES_E	Envelope element results
ENVRES_N	Envelope node results
ENVRES_I	Envelope integration station results
FATPROP	Fatigue properties
GEOM	Geometry of special element
HEAD	Heading describing the model
INISTR	Initial strains, displacements or rotations
MATERIAL [(2)]	Material property data
MOVE_GROUP	Move group during autostart
NOCOOR	Nodal coordinates of the model
NODPROP	Node interpolated element properties
NOORIENT	Orientation of nodes
PELOAD	External pressure and gravity loading
PILOAD	Internal pressure load
REEL	Reeling and straightening simulation scenario definition
THIST	Time history data
TIMECO	Time control data
TLOAD	Temperature loading
VISRES	Activate results for visual presentation by XPOST

[(1)] The properties for **BEAM** damping model are set at analysis start-up and cannot be changed during analysis restarts. For the **RALEIGH** damping model, the properties can be changed if the optional parameter **UPDATE=1** is applied, see Section 2.14 for further details.

[(2)] Material properties can be changed if the number of points in material curves is kept unchanged. This is convenient e.g. for seabed friction description in combination with autostart, see Section 2.6.

Linear material curves may be used during the start up, the real friction characteristic can be applied by using restart after start up, see Section 2.26.9. Note that during a restart all input data that do not influence the element and node definitions may be changed, i.e. all except the **CONTROL**, **NOCOOR**, **ELORIENT**, **ELCON**, **CONSTR** and **ELECC** commands.

In the following, the input data related to each identifier will be explained. The units are specified by using F = Force, M = mass, L = length, T = time, K = Temperature, E = Energy, R=radians. Square brackets ([ ]) are used to identify optional parameters.

The coordinate system is shown in Fig. 2.1. A right handed Cartesian coordinate system is applied. Gravity is applied in the negative global z-direction. If a sea surface

element is present in the model, this surface is assumed to be positioned at $z = 0$. Further, if a seabed contact surface is specified, this may be arbitrarily oriented and located in the BFLEX2010 model coordinate system by specifying the direction angle and (x0, y0) position vector relative to the BFLEX2010 coordinate system, see Section 2.8.

The local element end node coordinate system is shown in Fig. 2.2 where the element end is placed eccentric to the node. Positive eccentricities means that the element end is translated along positive axis relative to the element node. In the initial configuration the node coordinates system is parallel to the global system.

The definition of integration points related to the integration point results of tubulars and rectangles, see Section 2.38, is found in Fig. 2.3 and Fig. 2.4, respectively. The definition of corner stresses with reference to the old BFLEX raf file format is presented in Fig. 2.5

Note also that the definition of integration points around the pipe circumferencis are different in the old and new BFLEX format. For BFLEX2010 the first integration poiint P1 is located at the positive y-axis, ref. Fig. 2.18. For the old BFLEX raf format, P1 is defined at the negative z-axis.

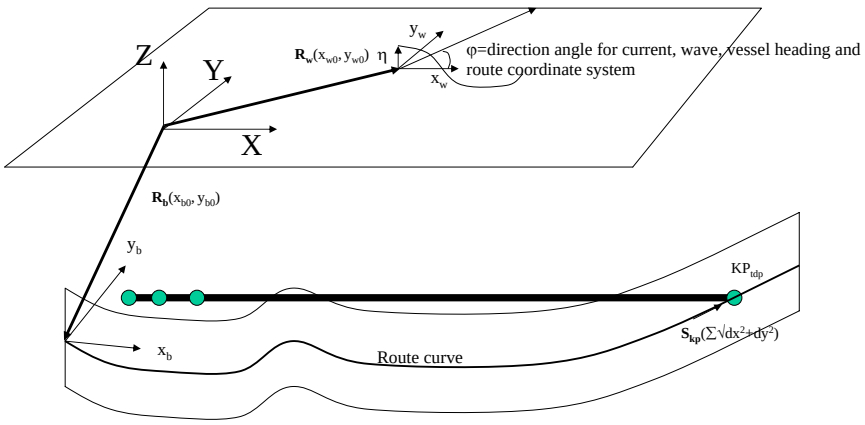


Figure 2.1: Coordinate system.

The model will always consist of nodes and elements. The orientation of each node is governed by the node coordinate system whereas the orientation of each element is

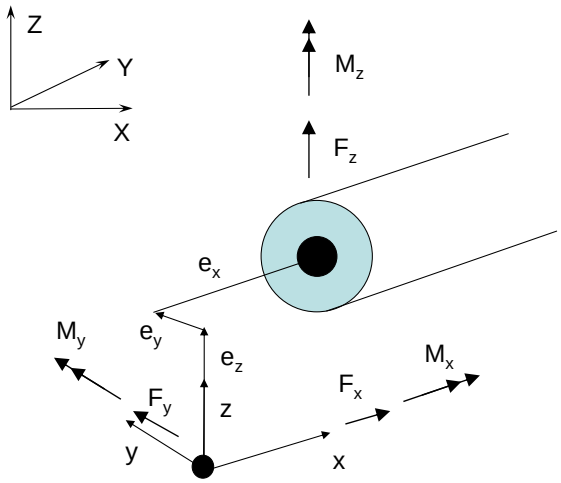


Figure 2.2: Coordinate system.

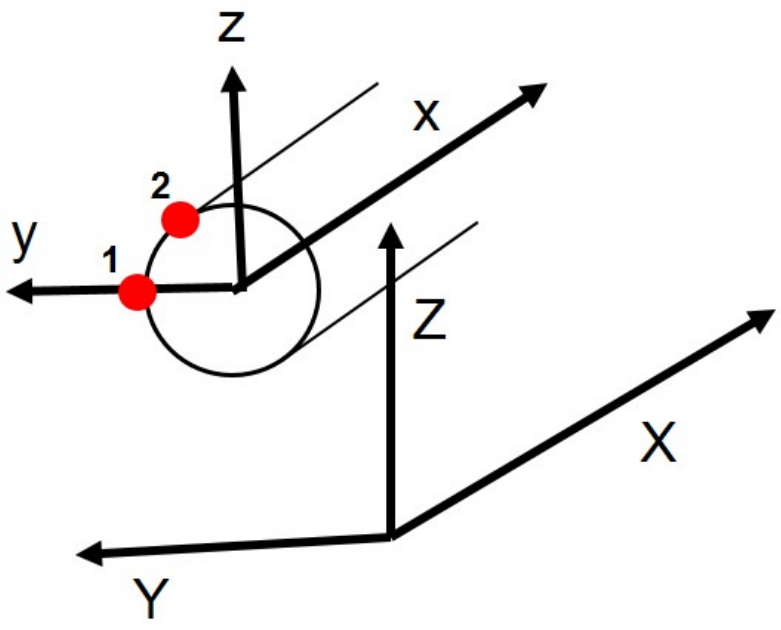


Figure 2.3: Definition of integration points for tubes

governed by the element coordinate system. As default the node coordinate system is

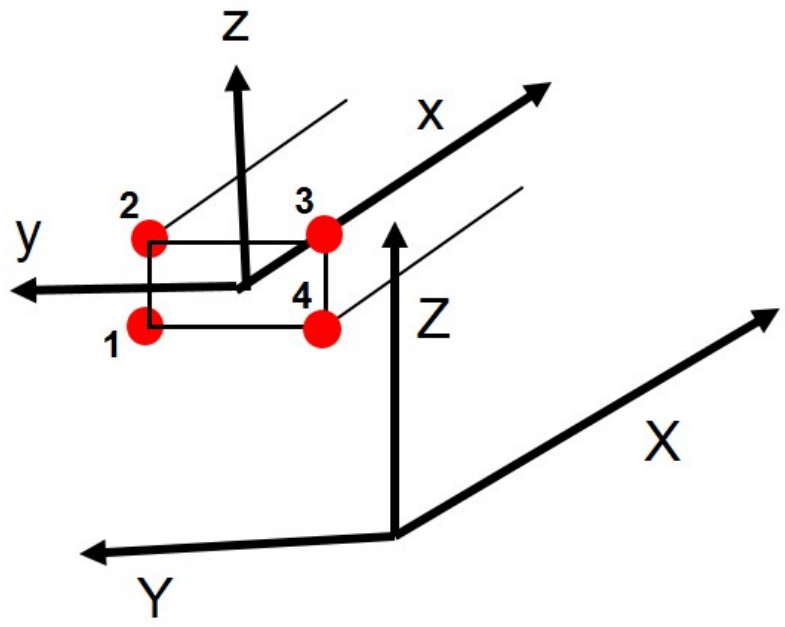


Figure 2.4: Definition of integration points for rectangles

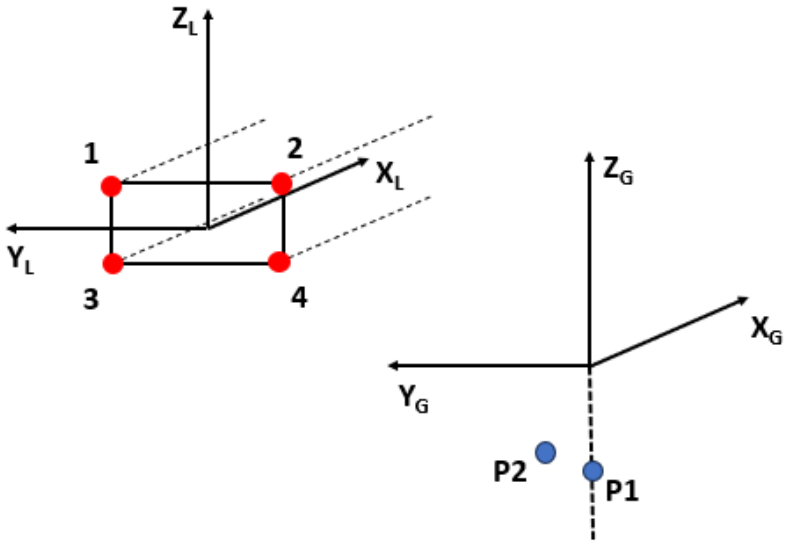


Figure 2.5: Definition of corners 1-4, and integration points (P1,P2) in the old raf file format

positioned parallel to the global coordinate system. However, the user may change that by the **NOORIENT** command, see Section 2.30.

The **REPEAT** command is used for several of the input cards. In all these cases the number of **REPEAT**s are the total number of instances created. Thus, if a REPEAT of 1 is applied, the **REPEAT** has no effect.

2.2 BONCON - Boundary Conditions

The boundary conditions may be specified as global or local boundary conditions. The following format is applied:

BONCON TYPE NODID DOF [MASTERNODE XANG YANG ZANG]
[REPEAT N NODINC]

where

TYPE: Type of boundary condition, which may have the values: **LOCAL**, **GLOBAL** or **SPECIAL**. **GLOBAL** and **LOCAL** means that the specified **DOF** is fixed in the global or local coordinates system, respectively. If the **SPECIAL** option is applied, the specified **DOF** is fixed in the local system of the **MASTERNODE**, or in a system with a specified rotation relative to the local **MASTERNODE** system.

NODID: Node ID number.

DOF: Nodal DOF number, where 1-3 is related to translation in xyz whereas 4-6 is related to rotation about xyz.

MASTERNODE: Master node number at which the applied boundary condition is referred. Only for **SPECIAL**.

XANG: Tait–Bryan angle around local x axis, applied to define the orientation of the applied boundary condition measured relative to the local system of the master node. Only for **SPECIAL** (unit: rad).

YANG: Tait–Bryan angle around local y axis, applied to define the orientation of the applied boundary condition measured relative to the local system of the master node. Only for **SPECIAL** (unit: rad).

ZANG: Tait–Bryan angle around local z axis, applied to define the orientation of the applied boundary condition measured relative to the local system of the master node. Only for **SPECIAL** (unit: rad).

The **REPEAT** command causes the previous sequence to be repeated N times:

N: Number of repeats.

NODINC: Node increment.

EXAMPLE:

```
#------------------------------------------------------------------------
# Boundary condition data
#------------------------------------------------------------------------
#
#      type    node   dof  repeat n nodinc
BONCON GLOBAL  1      1    REPEAT 3 2
```

```
BONCON GLOBAL  1      2
BONCON GLOBAL  1      3
```

The example above will put a boundary condition in direction 1 on nodes 1 3 5. Boundary conditions in directions 2 and 3 are active only for the first node.

2.3 CLOAD - Concentrated loads

The concentrated nodal loads are defined by the following format:

CLOAD HIST DIR NODE LOAD [NODE2 LOAD2]

where

HIST: Load history number.

DIR: Load direction (**1-3** = load along x-z, **4-6** = moment about x-z).

NODE: First node ID.

LOAD: Load magnitude for the first node.

NODE2: Last node ID.

LOAD2: Load magnitude for the last node. Linear load interpolation is applied for intermediate nodes.

The load components refer by default to the global coordinate system. The only exception is when a special reference system is used for the motion of the node. A special reference system will be applied in the following cases:

- When the node is assigned the boundary condition types **LOCAL** or **SPECIAL**, see the **BONCON** card in Section 2.2. Then the load components will refer to the local or special coordinate system of the node, which rotates together with the node during its motion.

- When the node is a helix node. This is the case if the node also is defined as element node number 3 or 4 for the HSHEAR352 and HSHEAR353 element types, see **NOD3** and **NOD4** defined by the **ELCON** card in Section 2.13. Then the load components will refer to the local coordinate system of the helix node.

To summarise:

DIR	Description
1	force along x-axis (unit: F)
2	force along y-axis (unit: F)
3	force along z-xis (unit: F)
4	moment about x-axis (unit: FL)
5	moment about y-axis (unit: FL)
6	moment about z-axis (unit: FL)

EXAMPLE:

```
#
# Concentrated nodal loads:
#-------------------------------------------
#     hist  dir  node  load
CLOAD 50    1    3001  1.18
#
```

2.4 CONSTR - Constraints

The **CONSTR** command allows the user to define constraints in the FE-model.

CONSTR CTYPE ...
[REPEAT ...]

The repeat card can be applied.

CTYPE: Type of constraint.

The following types of constraints can be specified:

PDISP : Prescribed displacement.

CONEQ : Constraint equation.

HISTCONEQ : Time dependent constraint equation.

For prescribed displacements the command has the following format:

2.4.1 PDISP

CONSTR PDISP PDTYPE NODID DOF ...
[REPEAT N NODINC]

PDTYPE: Type of prescribed displacement.

Allowed types are:

LOCAL : Apply prescribed displacement in local system of a node.

GLOBAL : Apply prescribed displacement in global system of a node.

Generally, the **REPEAT** command is used to repeat a sequence.

N: Total number of times the constraint is repeated.

NODINC: Nodal increment.

The other input data related to each option are defined below:

LOCAL

For **CONSTR PDISP LOCAL** the format is as follows:

```
CONSTR PDISP LOCAL NODID DOF DISPVAL HISTNO
[REPEAT N NODINC]
```

NODID: Node ID.

DOF: DOF number (1-6).

DISPVAL: Prescribed displacement value (unit: L for DOF 1-3, R for DOF 4-6). The dispacement value is multiplied with the time dependent load factor given in time history specified by the **HISTNO** parameter.

HISTNO: History ID, specifying the time dependent load pattern. See Section 2.35.

GLOBAL

For **CONSTR PDISP GLOBAL** the format is the same as for **LOCAL**:

```
CONSTR PDISP GLOBAL NODID DOF DISPVAL HISTNO
[REPEAT N NODINC]
```

NODID: Node ID.

DOF: DOF number (1-6).

DISPVAL: Prescribed displacement value (unit: L for DOF 1-3, R for DOF 4-6). The dispacement value is multiplied with the time dependent load factor given in time history specified by the **HISTNO** parameter.

HISTNO: History ID, specifying the time dependent load pattern. See Section 2.35.

EXAMPLE:

```
#       constrtype  type    nodid  dof  value   histno
CONSTR  PDISP       GLOBAL  1041   5    0.1745  200
```

2.4.2 CONEQ

A linear constraint equation has the general form: $r_{sl} = C_0 + C_1 r_{m1} + C_2 r_{m2} + C_3 r_{m3}$, where sl denotes slave and mi denotes the masters.

In the BFLEX2010-input a constant constraint equation with **CONEQ** option is specified by the following format:

```
CONSTR CONEQ PDTYPE ...
[REPEAT ....]
```

where:

PDTYPE: Type of contraint equation.

Allowed types are:

LOCAL : Apply constraints in local system of the slave and master nodes.
LOCSL : Apply constraints in local system of the slave node as reference.
GLOBAL : Apply constraints in global system of a node.

The other input data related to each option are defined below:

LOCAL

For **CONSTR CONEQ LOCAL** the format is as follows:

```
CONSTR CONEQ LOCAL SLNOD SLDOF C0 MNOD1 MDOF1 C1 ...
[REPEAT N SLAVEINC MASTINC]
```

SLNOD: Node ID of the prescribed (slave) node.
SLDOF: DOF number.
C0: Constant displacement (C_0).
MNOD1: ID for the master node.

MDOF1: Master DOF number.

C1: Constraint coefficient for the master number 1 (C_1).

The last parameters **MNOD1, MDOF1, C1** may be repeated k (number of masters) times.

The **REPEAT** card is described as follows:

N: Total number of times the constraint is repeated.

SLAVEINC: Nodal increment for slave.

MASTINC: Nodal increment for master(s).

LOCSL

For **CONSTR CONEQ LOCSL** the format is as follows:

```
CONSTR CONEQ LOCSL SLNOD SLDOF C0 MNOD1 MDOF1 C1 ...
[REPEAT N SLAVEINC MASTINC]
```

SLNOD: Node ID of the prescribed (slave) node.

SLDOF: DOF number.

C0: Constant displacement (C_0).

MNOD1: ID for the master node.

MDOF1: Master DOF number.

C1: Constraint coefficient for the master number 1 (C_1).

The last parameters **MNOD1, MDOF1, C1** may be repeated k (number of masters) times.

The **REPEAT** card is described as follows:

N: Total number of times the constraint is repeated.

SLAVEINC: Nodal increment for slave.

MASTINC: Nodal increment for master(s).

GLOBAL

For **CONSTR CONEQ GLOBAL** the format is as follows:

```
CONSTR CONEQ GLOBAL SLNOD SLDOF C0 MNOD1 MDOF1 C1 ...
[REPEAT N SLAVEINC MASTINC]
```

SLNOD: Node ID of the prescribed (slave) node.

SLDOF: DOF number.

C0: Constant displacement (C_0).

MNOD1: ID for the master node.

MDOF1: Master DOF number.

C1: Constraint coefficient for the master number 1 (C_1).

The last parameters **MNOD1, MDOF1, C1** may be repeated k (number of masters) times.

The **REPEAT** card is described as follows:

N: Total number of times the constraint is repeated.

SLAVEINC: Nodal increment for slave.

MASTINC: Nodal increment for master(s).

If all the constraint coefficients except for **C0** are zero, the **CONEQ** definition is equivalent to a **PDISP** definition. If all constraints coeffecient except for **C1** are zero, the **MDOF1** of **MNOD1** completely defines the behaviour of the **SLDOF** of **SLNOD**.

EXAMPLE:

```
#       ctype  type     slavnod  slavdof  disp  masternod  masterdof  constrcoeff
CONSTR  CONEQ  GLOBAL   1        1        0     1001       1          1.0
CONSTR  CONEQ  GLOBAL   1        2        0     1001       2          1.0
CONSTR  CONEQ  GLOBAL   1        3        0     1001       3          1.0
```

2.4.3 HISTCONEQ

A constraint equation with time dependent coefficients is specified by the following format:

```
CONSTR HISTCONEQ PDTYP SLNOD C0 THIST0 MNOD1 MDOF1 C1 THIST1
...
[REPEAT N SLAVEINC MASTINC]
```

where:

PDTYP: Type of constraint, can have values **LOCAL** or **GLOBAL**.

SLNOD: Node ID of the prescribed (slave) node.

SLDOF: Slave DOF number.

C0: Constant displacement (C_0) The dispacement value is multiplied with the time dependent load factor given in time history specified by the **THIST0** parameter.

THIST0: Time history number for C_0, refering to the definition of the time dependent load factor, see Section 2.35.

MNOD: ID for the master node.

MDOF: Master DOF number.

C1: Constraint coefficient for the master number 1 (C_1). The constrain coefficient value is multiplied with the time dependent load factor given in time history specified by the **THIST1** parameter.

THIST1: Time history number for C_1, refering to the definition of the time dependent load factor, see Section 2.35.

The sequence **MNOD1 MDOF1 C1 THIST1** can be repeated k (the number of masters) times.

The **REPEAT** card is described as follows:

N: Total number of time the constraint is repeated.

SLAVEINC: Nodal increment for slave.

MASTINC: Nodal increment for master(s).

EXAMPLE:

```
#
# Constraint equations between vessel and riser:
#-------------------------------------------------------
#                          sl   dof  C0   thist  ma  dof  C1  thist
CONSTR HISTCONEQ GLOBAL   101  3    0.0  100    55  3    1.0  200
```

2.5 CONTINT - contact interfaces

In order to optimize the contact search, the contact interfaces need to be defined. The following format is applied.

CONTINT GRPNAME MASTERNAME SLAVENAMEI [IS1 ISN TX TY TZ MAXIT IGAP]

where

GRPNAME: Name of the contact element group.

MASTERNAME: Name of the master element group.

SLAVENAMEI: Name of the slave element group or the name of a contact surface, see the **COSURFPR** command. If the element type of the contact element group is SEA150, then the master group shall have the same name as the sea group, and the slave group shall have the name of the structural element group that may have sea contact. For the SEA150 element type, no more parameters are needed. For structural elements or contact surfaces, additional parameters need to be given, see below.

IS1: First slave element in contact range. Dummy for HCONT and CONT152 elements.

ISN: Last slave element in contact range. Dummy for HCONT and CONT152 elements. Search for contact is carried out between **IS1** and **ISN**.

TX: Time at which the contact elements are to be activated in the local x direction.

TY: Time at which the contact elements are to be activated in the local y direction.

TZ: Time at which the contact elements are to be activated in the local z direction.

MAXIT: Gap iteration parameter which gives the maximum number of iterations to be performed in order to find whether there is a gap or not. If the number of iterations exceeds this number, gap is assumed for the remaining iterations. For CONT126 reasonable values is 4-6. For the other ones 20-60 are reasonable values.

IGAP: Control parameter used to control contact element features depending on contact element type.

For CONT164 the appropriate values are **IGAP** $\geq$ **0** or **IGAP** $<$ **0**. If **IGAP** $\geq$ **0** then the direction of the normal vector is set once at the start of the analysis. If **IGAP** $<$ **0** then the direction of the normal vector will be updated during the simulation as long as the distance between the master roller and the slave pipe is greater than zero.

For CONT126, **IGAP**=**2** means that the x-moment is set to zero and that the second term on the right-hand side of Eq. (2.11) is set to zero.

For CONT130 and CONT152 **IGAP** = **0** means that the feature is turned off.

For CONT130, **IGAP** $>$ **10** means that normal vector spline interpolation is applied in the longitudinal direction to obtain C^1-continuity between straight geometry segments. Spline interpolation gives improved accuracy for the axial force distribution along the slave system if the master is conical, eg. bellmouth (INSIDE = -1).

For CONT130, **IGAP** = **2** or **12** means that the torsion moment due to friction in the circumferential direction is not included. For **IGAP** = **12**, the torsion moment is not included while normal vector spline interpolation is applied in the longitudinal direction of the tube geometry.

For HCONT453, **IGAP**= 1 means that the friction force acting between layers is based on the concept of independent layers. This means that the direction of slip and associated friction forces are only depending on the motion of the relevant structural layer, i.e. no interaction of friction forces between layers is included. By default an isotropic friction model is used.

For CONT152, **IGAP** < **0** means that the meaning of igap transforms into time at which a new contact search is performed. For the CONT152 element the contact search is performed initially. If large motions occur this may lead to a large number of contact elements to ensure that contact is obtained at the right location. By this, the user can suppress a new contact search to update the candidate contact points.

EXAMPLE:

```
#
# Contact interface data:
#-------------------------------------------------------------------------
#        groupn    mname  sname          is1  isn    tx     ty  tz     gt1  gt2
CONTINT bscontact  core   bendstiffener  501  600  1000  10000   1      60    1
```

EXAMPLE:

```
#
# Contact interface data:
#-------------------------------------------------------------------------
#       grpname      mastername   slavename is1  isn  tx         ty    tz  maxit igap
CONTINT seabed       ormpipe1     cosurf1     1  201  1.      10000    1      6   1
CONTINT ormcontact   ormcontact   ormpipe2  341  441  10000   10000    1     50   1
CONTINT ormcontact1  ormcontact1  ormpipe2  341  441  10000   10000    1     50   1
CONTINT ormcontact2  ormcontact2  ormpipe2  341  441  10000   10000    1     50   1
CONTINT ormcontact3  ormcontact3  ormpipe2  341  441  10000   10000    1     50   1
CONTINT ormcontact4  ormcontact4  ormpipe2  341  441  10000   10000    1     50   1
CONTINT ormcontact5  ormcontact5  ormpipe2  341  441  10000   10000    1     50   1
CONTINT ormcontact6  ormcontact6  ormpipe2  341  441  10000   10000    1     50   1
CONTINT ormcontact7  ormcontact7  ormpipe2  341  441  10000   10000    1     50   1
CONTINT ormcontact8  ormcontact8  ormpipe2  341  441  10000   10000    1     50   1
CONTINT ormcontact9  ormcontact9  ormpipe2  341  441  10000   10000    1     50   1
CONTINT ormcontact10 ormcontact10 ormpipe2  341  441  10000   10000    1     50   1
CONTINT ormcontact11 ormcontact11 ormpipe2  341  441  10000   10000    1     50   1
CONTINT ormcontact12 ormcontact12 ormpipe2  241  441  10000   10000    1     50   1
CONTINT sea1         sea1         ormpipe1
```

2.6 CONTROL - CONTROL parameters

The control parameters are defined using one data card starting with **CONTROL** and in the following format:

```
CONTROL MAXIT NDIM ISOLVR NPOINT IPRINT CONR GAC ISTRES ...
```

where:

MAXIT: Maximum number of equilibrium iterations. This setting can be overwritten in the **TIMECO** card.

NDIM: Dimension of analysis:

- **3 :** 3-dimensional.

ISOLVR: Equation solver parameter;

- **1 :** gives skyline solver of the equation system.
- **2 :** gives sparse solver, which is most efficient.

NPOINT: Number of integration points around the cross section. This is used both for the pipe non-linear material elements and for the visual model meshing.

IPRINT: Print parameter One or two digits can be applied to set print of model information to output file (.bof), and print of timestep and iterations to log file (.blf) and print to screen. In particular print of time and iteration information can be timeconsuming.

- **0 :** turn off print to outputfile, logfile and screen
- **00 :** turn off print to outputfile, logfile and screen
- **01 :** turn off print to outputfile, turn on print to logfile and screen. Using **1** will also toggle this behaviour
- **02 :** turn off print to outputfile, print only timesteps and no information of convergence of iterations to logfile and screen
- **03 :** turn off print to outputfile, print timesteps and information of the converged iterations (last iteration) to logfile and screen
- **10 :** turn on print to outputfile, turn off print to logfile and screen
- **11 :** turn on print to outputfile, turn on print to logfile and screen
- **12 :** turn on print to outputfile, print only timesteps and no information of convergence of iterations to logfile and screen
- **13 :** turn on print to outputfile, print timesteps and information of the converged iterations (last iteration) to logfile and screen

CONR: Convergence norm. The recommended value is 10^{-5}–10^{-7}.

GAC: Acceleration of gravity (unit: LT^{-2})

ISTRES: Start procedure parameter which may have the following values: **STRESSFREE**, **RESTART**, **EIGEN** or **AUTOSTART**. **STRESSFREE** means that the initial configuration is stressfree. **RESTART** means restart from a previous analysis. **EIGEN** means eigenvalue analysis at a load step from a previous analysis.

The following parameters depend on the value given for **ISTRES**.

2.6.1 STRESSFREE

If **ISTRES** = **STRESSFREE**, no more parameters are required.

2.6.2 RESTART

If **ISTRES** = **RESTART** then one more parameter is needed:

CONTROL MAXIT NDIM ISOLVR NPOINT IPRINT CONR GAC RESTART IRESTP

IRESTP: Restart load step.

2.6.3 EIGEN

If **ISTRES** = **EIGEN** then two more parameters are needed:

CONTROL MAXIT NDIM ISOLVR NPOINT IPRINT CONR GAC EIGEN IRESTP NEVAL

IRESTP: Load step at which the eigenvalues are wanted.

NEVAL: Number of eigenvalues wanted. If a negative number is given, the eigenvalues are animated from irestp on the raf file

The load step **IRESTP** must be a step available at an existing raf-file with the same prefix name as the input file. Animation of the eigenmodes starting with the lowest mode is available on the raf-file after the analysis. The eigenvalues and the eigenvectors are printed to the output file. For the time being the eigenvalue analysis requires **ISOLVR**=1.

The number of accepted eigenvalues and the relative error for each eigenvalue are printed to the log file. The maximum number of eigenvalue iterations is for the time being set equal to twice the value of **NEVAL**. Hence, if the number of accepted eigenvalues is less

than the number of eigenvalues of interest, **NEVAL** must be increased such that the number of acceptable eigenvalues becomes large enough.

The computed eigenvalues and eigenmodes are valid for an undamped system, i.e. the system damping matrix is not accounted for in the computations.

EXAMPLE:

```
#-------------------------------------------------------------------------
# Control data
#
#         maxit ndim  isolvr  npoint  ipri  conr   gacc  iproc
CONTROL   100   3     2       16      1     1.e-5  9.81  stressfree
```

2.6.4 AUTOSTART

If **ISTRES** = **AUTOSTART** then an initial configuration is determined by BFLEX2010 automatically. It assumes that the pipeline is installed on a seabed defined by the **COSURFPR** card, see Section 2.8. It then finds the initial configuration by identifying candidate seabed contact points. A catenary section is included depending on lay vessel parameters. Note that the catenary section option **ICATEN** may be set to a non-positive integer which means that BFLEX2010 assumes the whole pipeline to be installed on seabed. The effective tension compatible with the initial configuration is computed by the following expression for submerged pipe elements:

$$T = \frac{w_s D}{\frac{1}{\cos\theta} - 1} \tag{2.1}$$

where D is the water column height above the pipe element, w_s is the pipe submerged weight and the departure angle is denoted θ. The departure angle must therefore be defined also for the case when the whole pipeline is located on the seabed (**ICATEN**$\leq$**0**). An external axial force with magnitude close to Eq. (2.1) should be applied at the end of the pipeline, otherwise the initial configuration may deviate too much from the static equilibrium configuration resulting in convergence difficulties at the first load step. Further, buoyancy and gravity loads are recommended to apply with full magnitude from time $t = 0.0$.

CONTROL MAXIT NDIM ISOLVR NPOINT IPRINT CONR GAC AUTOSTART IN1PIP IN2PIP INCPIP NROLLS ICATEN IVSNOD TB DEPAN FREEB RAMPAN RAMPLE STIRAD KPTDP0 SEABDGRP STINGERGRP VESSELGRP

The associated parameters are defined as follows:

IN1PIP: The first pipe element ID number at the start of the catenary.

IN2PIP: The last pipe element ID number at the vessel end.

INCPIP: The node increment along the element segment (normally 1).

NROLLS: Number of roller stations along stinger.

ICATEN: Catenary parameter..

- **2 :** then the catenary section is included, the seabed section is installed on seabed assuming small bending stiffness.
- **1 :** then the catenary section is included, the seabed section is installed on seabed assuming steel pipe bending stiffness.
- **0 :** no catenary section is included, the pipe is installed on seabed assuming steel pipe bending stiffness.
- **-1 :** no catenary section is included, the pipe section is installed on seabed assuming small bending stiffness.

Note that when setting **ICATEN**=**0**, or **ICATEN**=**-1** the pipe will be installed on seabed and BFLEX2010 will carry out a free-span analysis positioning last end of pipe at **KPTDP0**, see Fig. 2.6.

IVSNOD: Vessel pipe node ID (if relevant), i.e. the pipe node where the pipe is terminated in the tensioner. In order to move associated vessel model nodes the same quantity as the pipe node in order to reach the initial configuration it is necessary to give the vessel pipe node ID as input, see Fig. 2.7.

TB: Bottom tension (dummy) (unit: F).

DEPAN: Vessel departure angle (unit: R).

FREEB: Vessel freeboard (unit L). See Fig. 2.7

RAMPAN: Vessel ramp angle (unit: L).

RAMPLE: Vessel ramp length (unit: L).

STIRAD: Vessel stinger radius R (unit: L).

KPTDP0: Required initial KP value for TDP. Must be larger than 0.0.

SEABDGRP: Name of seabed element group.

STINGERGRP: Name of stinger contact element group (may be **NONE** for e.g. J-lay).

VESSELGRP: Name of vessel group (may be **NONE**). This parameter is used to identify whether there is a vessel and which elements that are used to represent the vessel. If not **NONE**, all nodes that are connected to element groups named 'vessel' for the first six characters will be translated the same distance as the pipe vessel node **IVSNOD**. The reference coordinate system applied is the sea bed coordinates taken from the seabed coordinate text file, see Section 2.8. In many cases, teese coordinates are based on sea bed surveys and may have large numric values, which may cause numerical difficulties. This may be solved by moving the origo of the sea bed survey data, see Section 2.8.

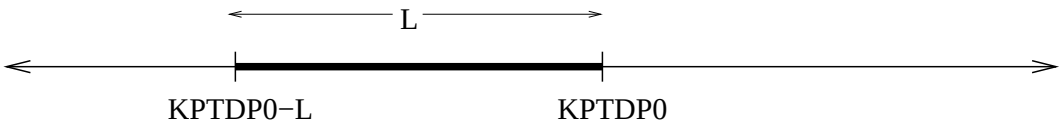


Figure 2.6: Position of pipe in free span analysis (**ICATEN** = **0**).

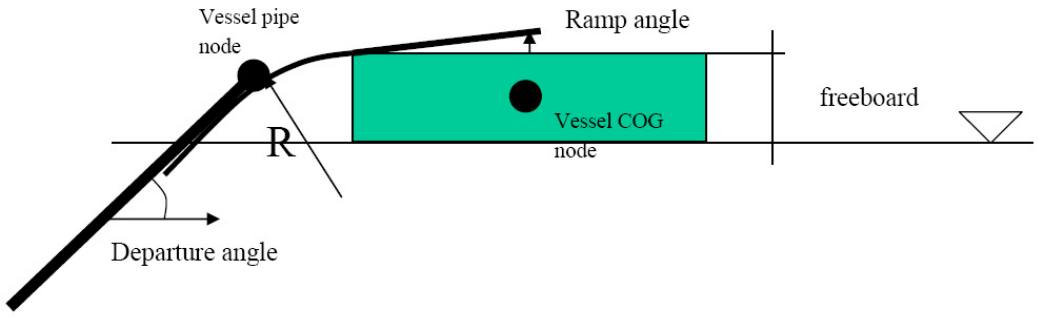


Figure 2.7: Vessel definition.

EXAMPLE:

```
CONTROL  120  3  1   16  1  1e-5  9.81  autostart
#
#       ie1pip  ie2pip  incpip nrolls  icaten ivsnod
        1       440     1      16      1        441
#
#       tens0  depang  freeb  rampan  rample   stirad kp
        0      1.244   10.0   0.1745  0        120    1
#
#       seabedgrp  stingergrp   vesselgrp
        seabed     ormcontact5  vessel1
#
```

2.7 COSUPR Contact surface material properties

The **COSUPR** command defines the material properties along a route/line on kilometer point (KP) basis. The command is required if a **COSURFPR** is defined. The following format is applied:

```
COSUPR MLINEID KP1 KP2 MNAME
                .. ..  ..
```

where:

MLINEID: Identity of the material line.

KP1: Starting point of material properties (KP value).

KP2: End point of material properties (KP value).

MNAME: Name of material for this KP range.

An arbitrary number of material sequences may be given.

The last parameters **KP1 KP2 MNAME** can be repeated as many times as needed to describe a varying material along the route. Note that **KP1** on a line must be the same as **KP2** of the preceeding line to ensure material to be defined at all locations.

The material line must be defined for the route section where contact between pipe and the contact surface may take place. The referenced material name must be defined with the **MATERIAL** command, see Section 2.26.

EXAMPLE:

```
#        route id    kp1     kp2        matname
COSUPR   100         -0.1    60000      soil1
```

2.8 COSURFPR - Contact surface properties

The **COSURFPR** command allows the user to define the contact surface properties, relative to the curvilinear position along the contact surface referred to as KP-points. The curvilinear length is measured on the route projected onto the xy-plane. The following format is applied:

COSURFPR CONAME COFILE NLINES KP0 XSTART YSTART ANGSTART MLINEID [.. MLINEIDN] [IDLINE]

where:

CONAME: Name of contact surface.

COFILE: Name of the ASCII datafile containing the seabed geometry description.

NLINES: Number of seabed lines in contact surface file. Must be an odd number 1,3,5 etc. A positive number of lines means that the data is on the standard format in Section 2.8.1.

KP0: KP-value for 1. point in contact surface datafile, i.e. the KP-value for the first data line in **COFILE**.

XSTART: x-coordinate at start of contact surface relative to the global coordinate system (unit: L).

YSTART: y-coordinate at start of contact surface relative to the global coordinate system (unit: L).

ANGSTART: Angular orientation of contact surface relative to the global coordinate system (unit: R).

MLINEID: Material ID of the route line, i.e. the center line if several lines are given.

MLINEIDN: Material ID of additional seabed lines. To be able to apply several seabed lines, they must also be defined in the contact surface file.

IDLINE: Optional. A number between 1 and **NLINE**, specifying that this seabed line is the only one to be used.

The first seabed line is the center line coinciding with the route line, and any succeeding seabed lines must alternate between the left and the right side of the center line as shown in Fig. 2.9. The pipeline will be placed along the seabed center line when using the AUTOSTART option in Section 2.6.4.

The material ID must be given for each of the seabed lines, but the same ID may be repeated for several lines. The material ID describes the material properties of the seabed line on a KP basis, and is defined by the **COSUPR** command, see Section 2.7.

The seabed lines should preferably be close to parallel with the pipeline. This is because the KP-value applied for determining the seabed material properties is computed based on the minimum distance between the pipeline node and the seabed lines.

Large coordinate input values should be avoided as they may give round-off errors in arithmetic operations. In such cases, it is beneficial to move and/or rotate the seabed coordinate system to reduce the coordinate values. This may be done by applying the **XSTART**, **YSTART** and **ANGSTART** parameters.

EXAMPLE:

```
#         name     data file      nline kpstart   x0        y0      fi        route id
COSURFPR cosurf1 "seabed.txt"    1     0         0         0       0         100
```

2.8.1 Standard route file

For a route file described by only a single seabed line, the format is as follows for each data line in **COFILE**:

x y z nx ny nz

repeated for as many points there are in the route. x, y and z are the coordinates of the point and nx, ny, nz are the global components of the sea bottom normal vector at

the point.

For a route file with several seabed lines, the x y z nx ny nz sequence is repeated for each seabed line in **COFILE** as shown in Fig. 2.8. The first sequence must be the center line, and should be followed by pairs of seabed lines located at each side of the center line, starting with the innermost pair, see Fig. 2.9. Note that BFLEX2010 computes the KP-value based on the seabed center line, also for the additional seabed lines.

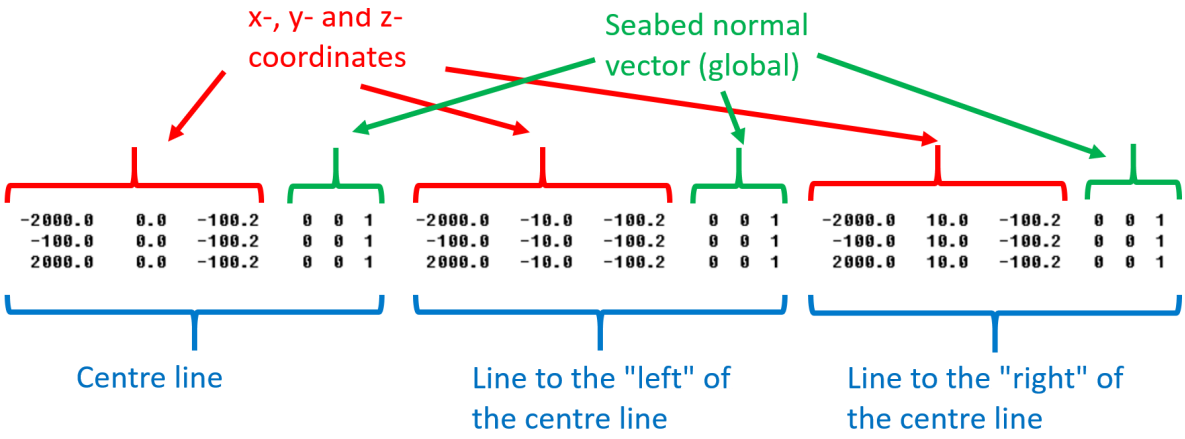


Figure 2.8: Input format for a route file with 3 lines.

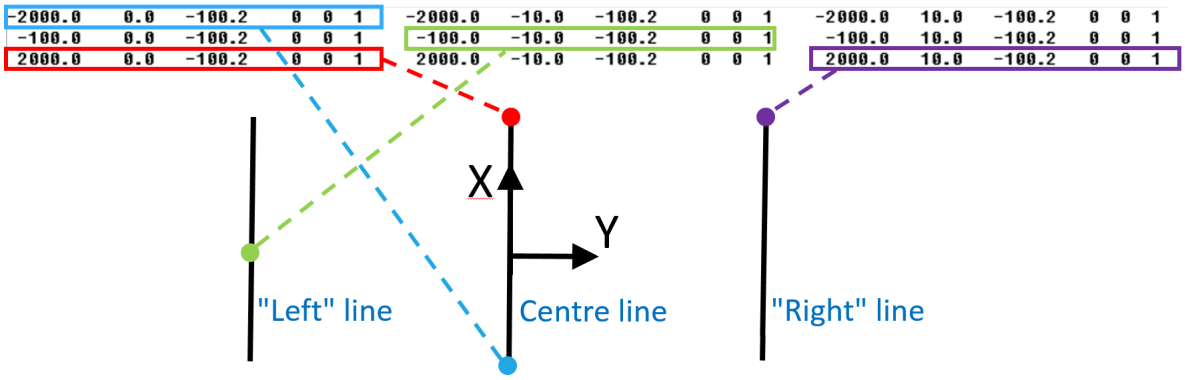


Figure 2.9: Illustration of line definition for a route file with 3 lines.

2.9 CROSSGEOM - Cross sectional geometry

The format of the card is as follows:

```
CROSSGEOM NAME CTYPE ....
```

where

NAME: name of cross sectional geometry. This will be used as reference in the **ELPROP** card, see Section 2.20.

CTYPE: Type of geometry for this section. The following types are implemented:

PIPE : Circular cross section.

BOX : Box shaped cross section.

GENERAL : General cross section .

BFLEX : BFLEX 2010 cross sections (carcass etc).

2.9.1 PIPE

For **CTYPE** = **PIPE** the format is:

```
CROSSGEOM NAME  PIPE RM TH
```

where:

RM: Mean radius. (unit: L)

TH: thickness. (unit: L)

2.9.2 BOX

For **CTYPE** = **BOX** the format is:

```
CROSSGEOM NAME  BOX WI HI WO HO
```

where:

WO: Outside width of box. (unit: L)

HO: Outside height of box. (unit: L)

WI: Inner width of box. (unit: L)

HI: Height inside of box. (unit: L)

If **CTYPE** = **GENERAL**, the geometry is defined by a local right handed Cartesian xyz-coordinate system as shown in Fig. 2.12. The local x-axis points along the pipe.

The geometry is defined by boundary curves where the boundary is divided into a number of segments which again may be divided into an arbitrary number of intervals **NINTER**. The geometry of the boundary curves is always to be specified in counter-clockwise order about the local x-axis.

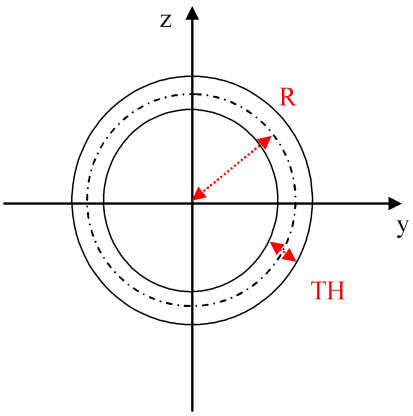


Figure 2.10: **CROSSGEOM** of type **PIPE**.

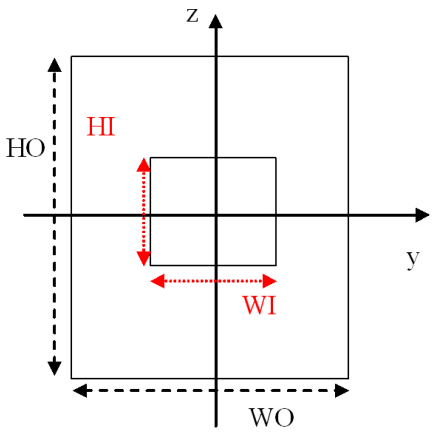


Figure 2.11: **CROSSGEOM** of type **BOX**.

2.9.3 GENERAL

For **CTYPE** = **GENERAL** the format is:

CROSSGEOM NAME GENERAL Y0 Z0 CURVCODE P1 P2 P3 NINTER INTERFACE
... **...** **...**

Y0: In order to describe the geometry of hollow section BEAM elements it is necessary to use two boundary curves. **Y0** defines the y-start point of the inner boundary curves relative to where the outer boundary curve starts.

Z0: In order to describe the geometry of hollow section BEAM elements it is necessary to use two boundary curves. **Z0** defines the z-start point of the inner boundary curves relative to where the outer boundary curve starts.

CURVCODE: Curve code, given as a character string which may have the following

values:

S : Segment is straight

CI : Segment is circular with the circle centre at the cross section inside.

CO : Segment is circular with the circle centre at the cross section outside.

P1: If **CURVCODE** = **S**: length of segment, unit: mm. If **CURVCODE** = **CI** or **CO**: angle at start of circle, unit: deg.

P2: If **CURVCODE** = **S**: angular orientation of segment local YZ system, where the Y-axis is directed along the element and the Z-axis is pointing outwards, unit: deg. If **CURVCODE** = **CI** or **CO**: angle at end of circle, unit: deg.

P3: Circle radius. Note that when specifying shells, the radius is always the outer radius, unit: mm. Dummy for **CURVCODE** =**S**.

NINTER: Number of intervals within the segment.

It is preferred that the boundary is defined with nodes exactly on the top, at the bottom and at the leftmost and rightmost location of the geometry. This will minimize unintended effects from constraints that are applied to obtain stability.

INTERFACE: The local interface number. Interface numbers higher than 1 are used to describe holes in a geometry.

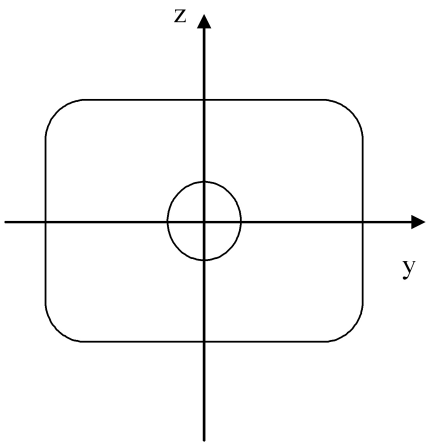


Figure 2.12: **CROSSGEOM** of type **GENERAL**.

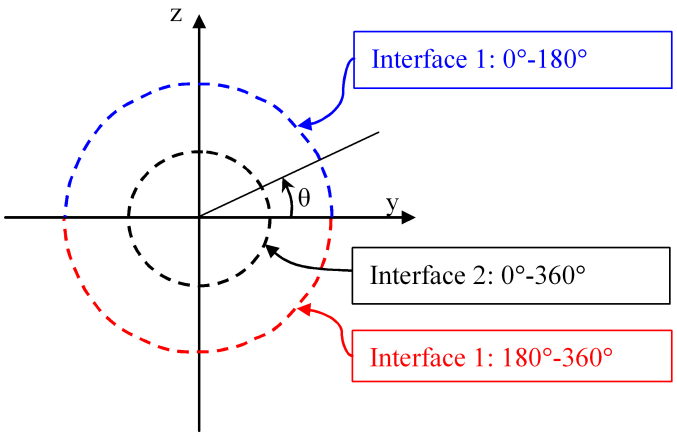


Figure 2.13: **CROSSGEOM** *pipegen* of type **GENERAL** from example.

EXAMPLE:

```
#           name      ctype    WO     HO    WI   HI
CROSSGEOM   boxsec    BOX      0.5    0.5   0.4  0.4
#           name      ctype      R     TH
CROSSGEOM   pipesec    PIPE     0.45    0.1
#           name      ctype    Y0  Z0  CURVCODE P1   P2   P3   NINTER INTERFACE
CROSSGEOM   pipegen   GENERAL   1   0    CI       0    180   1. 30       1
                             -1   0    CI     180    360   1. 30       1
                            0.5   0    CI       0    360   .5 30       2
#
```

2.9.4 BFLEX

For **CTYPE** = **BFLEX** the format is:

CROSSGEOM NAME BFLEX Y0 Z0 CCURVE P1 P2 P3 P4 NINTER ICODE
... ...

Y0: The start local cross-section y-coordinate of the cross section (unit: L).

Z0: The start local cross-section z-coordinate of the cross-section (unit: L).

CCURVE: Curve code, given as a character string of length 2. The legal options are:

- **S :** Straight segment.
- **CI :** Circular segment with the circle centre at the cross section *inside*.
- **CO :** Circular segment with the circle centre at the cross section *outside*.

P1: Length or angle. If **CCURV** = **S**, **P1** = length of segment (unit: L). If **CCURV** = **CI** or **CO**, **P1** = angle at start of circle (unit: deg).

P2: Angular orientation or angle at end of circle. If **CCURV** = **S**, **P2** = angular orientation of segment local YZ system, where the Y-axis is directed along the element and the Z-axis is pointing outwards (unit: deg). If key CCURV = **CI** or **CO**, **P2** = angle at end of circle (unit: deg).

P3: Circle radius (dummy for **CCURV** = **S**, unit: L).

P4: Thickness of segment (only applicable for **CCODE** = **CARC**. Unit: L).

NINTER: Number of intervals within the segment.

ICODE: Segment code. The legal options are:

- **0 :** The segment has no nub/valley contact and is not exposed to direct loading by pressure
- **1 :** The segment is exposed to pressure load (normally part of the inner surface and parts of the outer surface interacting with tensile armour only)
- **2 :** Nub contact segment, see Fig. 2.15.
- **3 :** Valley contact segments, see Fig. 2.15.

Note that for a zeta spiral, a segment of itcode 0 is required between a switch between the contact codes 1,2 and 3. Also, the first segment should have itcode 0.

Comment: The cross section is defined by a local right handed Cartesian xyz coordinate system which is related to the global pipe coordinate XYZ system as shown in Fig. 2.14 The X-axis points along the tendon assuming the tendon to be installed by a positive lay angle with respect to geometric orientation. The applied axis system is shown in Fig. 2.14.

The geometry of the boundary curve is then given in counterclockwise order as described above.

The definition of diameter using key CCURV = **CI** or **CO** are shown in Fig. 2.16.

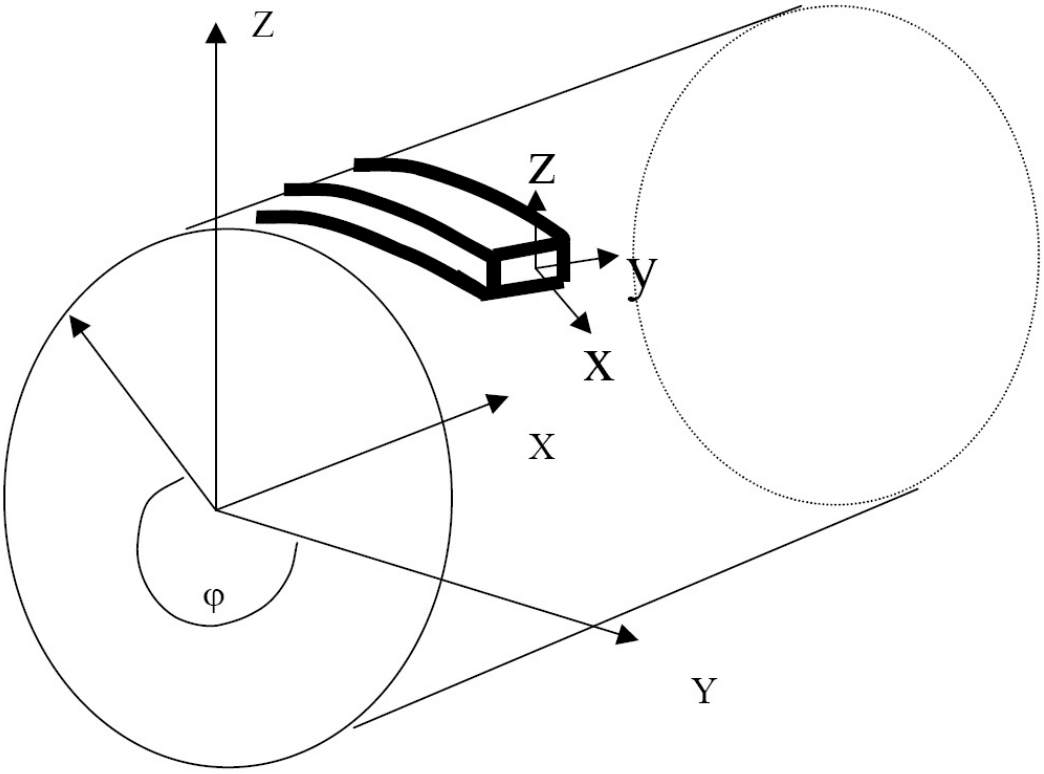


Figure 2.14: Axis systems.

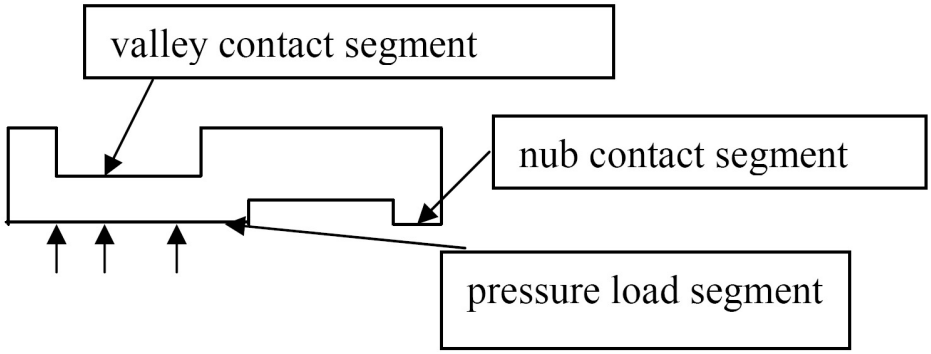


Figure 2.15: Segment description.

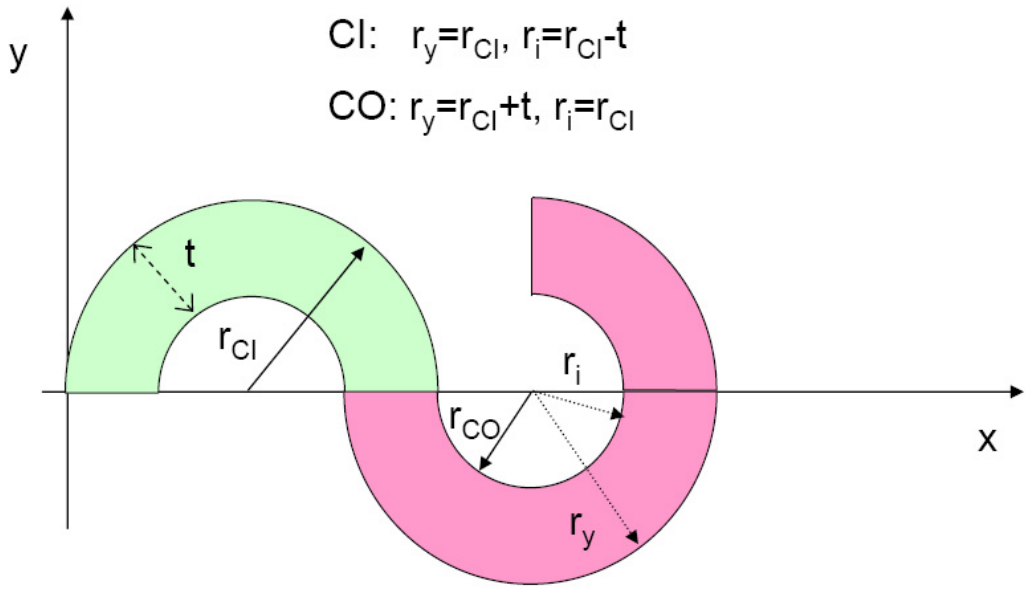


Figure 2.16: **CROSSGEO** of type **BFLEX**.

EXAMPLE:

```
# simple example
#              name    ctype    y0    z0  ccurv      p1          p2        p3        p4    ninter     icode
CROSSGEOM CCLI-SIRK   BFLEX     0     0     CI      0.0000      360.     1.000    0.0          10         0
#
#
```

2.10 CROSSECTION- BFLEX2010 Cross sectional input

The format of the card is as follows:

CROSSECTION MYPIPENAME CTYPE ....

where

MYPIPENAME: Name of BFLEX2010 cross sectional geometry.
CTYPE: Type of geometry for this section. The following types are implemented:

FLEXCROSS :
353FLEXCROSS :
BENDSTIFF :
NLBENDSTIFF :
BENDSTIFF-BEND :
NLBENDSTIFF-BEND :

A flexible pipe cross-section can be built in three different ways:

1. By using FLEXCROSS as described below and defining a core element group using PIPE52 and tensile armour groups using PIPE52. PIPE52 is a beam element. The axial, torsion and bending stiffness of the core is established from all layers except the tensile armours. For the tensile armour layers the non-linear moment curvature behaviour is established on the basis of a selected time along the load history (normally at the time the axisymmetric loads have been given its full value). A linear relationship is established for the torsion and axial stiffness. Note that the contact pressure is updated at each step using an algorithm by which gaps due to external pressure is taken into account throughout the analysis.

2. By using FLEXCROSS as described below and defining a core element group using PIPE52 and tensile armour groups using HSHEAR352. For the core, the same as described above applies. HSHEAR352 is a sandwich beam element that only allows for longitdinal slip and requires to be defined by a number of helices in each layer

using polar coordinates, see Section 2.28(To have a 16 point resolution around the circumference, 16 helices are required). The stick-slip behaviour between layers is eastablished as a shear stress - relative deformation characteristic on the basis of a selected time along the load history (normally at the time the axisymmetric loads have been given its full value). A linear relationship is established for the torsion and axial stiffness. Note that the contact pressure is updated at each step using the same algorithm as for the above.

3. By using 353FLEXCROSS as described below allowing all layers of a flexible pipe to be modelled using HSHEAR363 or HSHEAR364 for the pressure spiral, carcass, anti-buckling tape and plastic layers, HSHEAR353 for the tensile armour group. HSHEAR363 and HSHEAR364 are combined beam and shell elements that describes the local radial motion in an approximate way. The difference between them is that HSHEAR364 also includes the through thickness strain (thick shell). HSHEAR353 is a beam element that allows for both transverse and longitudinal slip and which requires the tensile armour to be defined by a number of helices in each layer using polar coordinates, see Section 2.28 (To have a 16 point resolution around the circumference, 16 helices are required). The properties of the structural layers are established from the cross-section input given as specified below. Layer contact between the different layers requires several contact element groups using HCONT463 to desribe contact between HSHEAR363/HSHEAR364 and HSHEAR353 elements. Then HCONT464 can be applied to both describe contact between HSHEAR363 and HSHEAR364 layers. The contact and stick-slip behaviour can be modelled manually by specifying a the material card associated to HCONT463 or HCONT453 or automatic by the 353FLEXCROSS feature described here on the basis of the contact pressure at a selected time along the load history (normally at the time the axisymmetric loads have been given its full value). It is noted that it is not allowed to combine layers of HSHEAR363 (no through thickness strain) and HSHEAR364 (through thickness strain included). If the layers, other than tensile armour layers, are described by HSHEAR364, the thickwall assumption is adopted and through thickness strain is also included for the tensile armnour (the HSHEAR353 elements). This is made possible by utilizing the two dummy torsion DOFs at each element end. Then these must not be suppressed by applying a boundaty condition, which is mandatory in the thin shell assumption because then the torsion DOF is dummy..

2.10.1 FLEXCROSS

For **CTYPE** = **FLEXCROSS** the format is:

CROSSECTION MYPIPENAME FLEXCROSS IFRIC DISFAC FORFAC GEOFAC ENDFAC DI TIMEINI ITCODE ILAEXT IELBFL FIMOD CONTDEN NELGR EL_1GRP EL_2GRP ... EL_NGRP
CTYPE TH MATNAME FRIC LAYANGLE RNUM TEMP MATNAME CCODE CFATFL AREA IT INY IKS WIDTH [PRLOC]
CTYPE ....
CTYPE ....

where:

IFRIC: Solution algorithm code for handling friction.

- **1 :** Only friction from axisymmetric effects is included when solving equilibrium equations.
- **2 :** The friction is updated by the current contact stress obtained during bending the pipe.

DISFAC: Maximum relative displacement in shear spring (last point on shear spring material curve, see old Bflex).

FORFAC: Maximum non dimensional force (last point on shear spring material curve, see old Bflex).

GEOFAC: Bending stress calculation factor

- **0 :** Loxodromic, see (Sævik, 1992).
- **1 :** Geodesic, see (Sævik, 1992).

ENDFAC: end shear spring stiffness factor for element type HSHEAR352. By setting a negative value, all elements will have the same friction spring characteristis. The boundary condition at the helix end nodes must be suppressed in the 1 direction to simulate the end fitting by using **LOBON**. Note that the physical meaning of the local 1 DOF is *core relative displacement* (only applicable for element type HSHEAR352)

DI: Inner diameter

TIMEINI: This is the time when the stick-slip properties of the friction springs are calculated. NB! The value *must* be larger than the time step at which the tension and pressure sequence has been completed, but less than the first time step with bending loads. This normally requires at least one time step without loading.

ITCODE: This is a code specifying which solution algorithm to be used. It may have the following values:

- **0 :** Full equilibrium iteration of the entire cross-section at each load step, slip value found from inner layer (ref. itcode 21).

1 : Full equilibrium iteration of the entire cross-section at each load step, slip value found for each layer (ref. itcode 31).

21 : The moment contribution from the tendons is taken into account by a friction moment approach. The stresses are calculated by iteration with respect to moment balance (same as for old bflex version before BFLEX2010). One moment-curvature curve is applied for all layers, slip is governed by the inner layer.

31 : The moment contribution from the tendons is taken into account by a friction moment approach as for **ITCODE 21**. However, **ITCODE 31** applies one curve for each layer so that the slip process is more correctly modelled. The moment curvature relation before start slip assumes plane surfaces remain plane. Between start slip and full slip a numerical integration procedure is used to give a best fit with respect to measured moment curvature data. The friction stresses are calculated from the resulting friction moment and has proven to give less stress in extreme cases than **ITCODE 0**. However, very good correlation was found when comparing against FBG full scale test stress data with respect to fatigue.

32 : The moment contribution from the tendons is taken into account by a friction moment approach as for **ITCODE 31**, where one curve is applied for each layer. The moment curvature relation before slip and between start slip and full slip is based on assuming an infinite stiff core. This is the option that should be used when introducing the shear interation parameter, see Section 2.20.14

ILAEXT: The interface number (outside outer layer = number of layers) where external pressure is applied. By specifying a negative number equal to the number of cross-section layers, the user might specify the local pressure (prloc) as an extra parameter for each layer

IELBFL: Element where BOUNDARY/PFLEX model is studied(located)

FIMOD: Angle position of BOUNDARY model (unit: deg)

CONTDEN: Density of content (unit: $ML^{-}3$)

NELGR: Number of element groups related to core and tension armour layers

EL1GRP: Name of element group 1. Must be a PIPE52 group representing the core of the tensile pipe, Section 2.13.

EL2GRP: Name of element group 2. Representing tensile armour layer 1. Either a PIPE52 group for **ITCODE21** and **31**, or a HSHEAR group for **ITCODE0**, Section 2.13.

ELNGRP: Name of element group N representing tensile armour layer N.

CTYPE: Cross-section layer type identifier (character string of length 4). Legal values are:

CARC : Carcass

THER : Thermoplastic layer

ZETA : Zeta type pressure spiral

SPIR : Pressure spiral

THET : Theta type pressure spiral

TCLI : Clip used together with theta

CCLI : C-clip

TENS : Tensile armour

For the different types, some of the input data are dummy (see description below).

TH: Layer thickness (unit: L).

MATNAME: Material name.

FRIC: Friction coeffcient.

LAYANGLE: Lay angle (unit: deg).

RNUM: Number of tendons. (Dummy for **CTYPE=THER**).

TEMP: Temperature in layer (unit: T).

NLMAT: Non-linear material curve name.

CCODE: Cross section control code, which may have the following values (character string with min. 7 and max. 12 letters):

A : MANUAL

B : An arbitrary name of the cross section (character string with min. 7 and max. 12 letters) that need to be defined under the optional data group **CROSS-SECTION BOUNDARY DATA**

Dummy for **CTYPE=THER**.

CFATFL: The name of the data file containing the fatigue data to be used (character string of max. length 12). Dummy for **CTYPE=THER**.

AREA: Cross section area of tendon. Dummy for **CTYPE=THER**, and if **CCODE≠MANUAL**. (Unit: L^2).

IT: Torsional inertia moment of tendon. Dummy for **CTYPE=THER**, and if **CCODE≠MANUAL**. (Unit: L^4).

INY: Inertia moment about strong axis of tendon. Dummy for **CTYPE=THER**, and if **CCODE≠MANUAL**. (Unit: L).

IKS: Inertia moment about weak axis of tendon. Dummy for **CTYPE=THER**, and if **CCODE≠MANUAL**. (Unit: L^4).

WIDTH: Width of tendon. Dummy for **CTYPE=THER**, and if **CCODE=MANUAL**. (Unit: L).

PRLOC: The local pressure on the inside of the layer. Optional and requires that **ILAEXT** is set to the maximum number of layers with a negative value i.e. -NLAYER. (Unit: FL^{-2})

Note: If **NLMAT**≠**NONE**, material type for this group must be ELASTOPLASTIC.

Note: If **CCODE**=**MANUAL** the remaining cross-section parameters must be specified by the user.

Note: If the cross-section is to be defined under the **CROSS-SECTION BOUNDARY DATA** group, the program will calculate the cross-section data based on the boundary curve given. The remaining data are therefore dummy in this case.

Note: The contents of the file **CFATFL** are described in Section 2.10.7.

Note: Internal pressure is loaded by **PILOAD** on the PIPE52 elements, and will be applied to the inner surface of the innermost cylidrical layer (**CTYPE**=**THER**).

2.10.2 353FLEXCROSS

For **CTYPE** = **353FLEXCROSS** the format is:

```
CROSSECTION MYPIPENAME  353FLEXCROSS DI TIMEINI ILAINT ILAEXT
IELBFL FIMOD CONTDEN NLAYGRP EL1GRP ... ELNGRP
CTYPE TH MATNAME FRIC LAYANGLE RNUM TEMP MATNAME CCODE
CFATFL AREA IT INY IKS WIDTH
CTYPE ....
CTYPE ....
```

where:

DI: Inner diameter

TIMEINI: This is the time where the stick-slip properties of the friction springs are calculated. The value *must* be larger than the time step at which the tension and pressure sequence has been completed and less than the first time step with bending loads. This normally requires at least one time step without loading. If a negative value is given, the user defined properties will be used throughout the analysis.

ILAINT: The interface number (inside inner layer = 1) where internal pressure is applied

ILAEXT: The interface number (outside outer layer = number of layers) where external pressure is applied.

IELBFL: Element where BOUNDARY/PFLEX model is located (dummy as per now as the BOUNDARY/PFLEX functionality only exist for FLEXCROSS, see Section 2.10.1)

FIMOD: Angle position of BOUNDARY model (dummy as per now as the BOUNDARY/PFLEX functionality only exist for FLEXCROSS, see Section 2.10.1) (unit: deg)

CONTDEN: Density of content

NLAYGRP: Number of layer element groups applied to describe the cross-section

EL1GRP: This is a text string to be specified either as **LAY1-LAY2-ELGRPNAME1-ELGRPNAME2** or **LAY1-LAY2-ELGRPNAME1**. **LAY1** and **LAY2** refers to the cross-section layer number starting from the inside. If one ELGRPNAME reference is given it must refer to a structural element group (either HSHEAR353 for the tensile armour or HSHEAR363/HSHEAR364for the other layers) defined to represent the layer, see Section 2.13. If two element groups are given, one of these groups has to be a contact element group using either of HCONT453, HCONT454, HCONT463 and HCONT464. Note that HCONT453 is applied to model direct contact between two tensile armours modelled by HSHEAR353, whereas HCONT463 is applied to model contact between a plastic layer modelled by HSHEAR363 or HSHEAR364 and a tensile armour layer modelled by HSHEAR353. The former alternative will normally give smaller friction stresses due to bending because the the direction of the friction forces will result from the slip directions of both layers, whereas for the latter alternative the relative motions between layers will be uncoupled and the dominating friction component will be directed in the longitudinal helix direction during bending. HCONT464 is applied to model contact pressure loads between layers of HCONT364 elements. HCONT454 is applied to address layer contact in the hoop direction between two consequitive wires modelled by HSHEAR353.

The friction coefficient will be taken from the inside of the next structural layer relative to the **LAY2** specified. In order to calculate the contact element stiffness the layer Young's modulus is used in the following way: If the first reference **ELGROUPNAME1** is a contact element group, this will be taken from the next layer. In the opposite case this will be taken from the current layer. The layer numbering starts from the inside (**LAY1**=1=normally carcass) For the pressure spiral and plastic/tape layers, corresponding HSHEAR363 structural groups need to be specified, whereas for each tensile armour layer a HSHEAR353 group is needed, see Section 2.13. It is possible to merge subsequent plastic layers to be modelled by HSHEAR363 into one element group. However, no merging of spiral/tape layers is allowed and they need to refer to separate element groups. This is because a plastic layer modelled by HSHEAR363 is represented as a two-dimensional thin shell whereas in the spiral/tape layer case a one -dimensional

representation is applied that only includes the axisymmetric strains related to axial, torsion and radial motions possibly having differnt lay angles. The latter case will therefore require separate structural element groups to be defined. The anti-buckling tape is to be considered as a spiral layer. In case of merging, the mechanical properties in terms of axial, torsion, bending and radial stiffness will be the sum of all layers. Between layers where interfaces are deemed necessary to handle gap effects, a contact group is needed. If no contact group is specified, then the same node system must be used for the corresponding elements. It is recommended to avoid *hard contact interfaces, i.e. contact elements between metal layers (if not separated by tape layers) to avoid numerical problems.*

CTYPE: Cross-section layer type identifier (character string of length 4). In addition to the same layer types for FLEXCROSS, a TAPE representation is allowed for in 353FLEXCROSS:

TAPE : Anti buckling tape, same syntax as for tensile armour.
CARC : Carcass
THER : Thermoplastic layer
ZETA : Zeta type pressure spiral
SPIR : Pressure spiral
THET : Theta type pressure spiral
TCLI : Clip used together with theta
CCLI : C-clip
TENS : Tensile armour

For the different types, some of the input data are dummy (see description below).

TH: Layer thickness (unit: L).

MATNAME: Material name.

FRIC: Friction coeffcient.

LAYANGLE: Lay angle (unit: deg).

RNUM: Number of tendons. (Dummy for **CTYPE=THER**).

TEMP: Temperature in layer (unit: T).

NLMAT: Non-linear material curve name.

CCODE: Cross section control code, which may have the following values (character string with min. 7 and max. 12 letters):

A : MANUAL
B : An arbitrary name of the cross section (character string with min. 7 and max. 12 letters) that need to be defined under the optional data group **CROSS-SECTION BOUNDARY DATA**

Dummy for **CTYPE=THER**.

CFATFL: The name of the data file containing the fatigue data to be used (character string of max. length 12). Dummy for **CTYPE=THER**.

AREA: Cross section area of tendon. Dummy for **CTYPE=THER**, and if **CCODE≠MANUAL**. (Unit: L^2).

IT: Torsional inertia moment of tendon. Dummy for **CTYPE=THER**, and if **CCODE≠MANUAL**. (Unit: L^4).

INY: Inertia moment about strong axis of tendon. Dummy for **CTYPE=THER**, and if **CCODE≠MANUAL**. (Unit: L).

IKS: Inertia moment about weak axis of tendon. Dummy for **CTYPE=THER**, and if **CCODE≠MANUAL**. (Unit: L^4).

WIDTH: Width of tendon. Dummy for **CTYPE=THER**, and if **CCODE=MANUAL**. (Unit: L).

Note: If **NLMAT≠NONE**, material type for this group must be ELASTOPLASTIC.

Note: If **CCODE=MANUAL** the remaining cross-section parameters must be specified by the user.

Note: If the cross-section is to be defined under the **CROSS-SECTION BOUNDARY DATA** group, the program will calculate the cross-section data based on the boundary curve given. The remaining data are therefore dummy in this case.

Note: The contents of the file **CFATFL** are described in Section 2.10.7.

Note: 353FLEXCROSS only supports the tensile armour model and will not build and activate the PFLEX/BOUNDARY/LIFETIME features.

Note: **ILAEXT** and **ILAINT** can not be identical.

EXAMPLE:

```
#            NAME    TYPE         DI       TIMEINI ILAINT ILAEXT IELBFL FIMOD CONTDEN NLAYGRP
CROSSECTION  mypipe  353FLEXCROSS 101.6e-3 1.1     2      9      1      0     1000    10
# EL1GRP ... ELNGRP
1-1-carcass
3-3-zeta
2-4-seal-contactseal
5-5-tapeoutwardcontact-tensile1
5-5-interlayercontact1-tensile1
6-6-strutape-tapeinwardcontact
7-7-sheathcontact-tensile2
7-7-interlayercontact2-tensile2
8-8-outersheath
9-9-bucklingtape
#
# CTYPE   TH  MATNAME    FRIC LAYANG  RNUM TEMP NLMAT CCODE  CFATFL AREA    IT  INY IKS WIDTH
```

```
CARC  5.0e-3  steel_316  0.01 87.828  1    0.0  none  MANUAL NONE   36.0e-6 0.0 0.0 0.0 0.0
THER  5.1e-3  plast_PVDF 0.01 0.000   0    0.0  none  NONE   NONE   0.00    0.0 0.0 0.0 0.0
ZETA  6.4e-3  steel_110  0.01 87.813  1    0.0  none  MANUAL NONE   79.2e-6 0.0 0.0 0.0 0.0
THER  2.0e-3  plast_PA11 0.01 0.000   0    0.0  none  NONE   NONE   0.00    0.0 0.0 0.0 0.0
# 1st armour
TENS  2.0e-3  steel_190  0.01 -38    61    0.0  none  FLEXT  NONE   0.00    0.0 0.0 0.0 0.0
THER  2.0e-3  plast_PA11 0.01 0.000   0    0.0  none  NONE   NONE   0.00    0.0 0.0 0.0 0.0
# 2nd armour
TENS  2.0e-3  steel_190  0.01 38     65    0.0  none  FLEXT  NONE   0.00    0.0 0.0 0.0 0.0
THER  6.0e-3  rubber     0.01 88.000  0    0.0  none  NONE   NONE   0.00    0.0 0.0 0.0 0.0
TAPE  1.0e-3  glass_fil  0.01 83.5    1    0.0  none  MYTAPE NONE   0.00    0.0 0.0 0.0 0.0
#
#
```

2.10.3 BENDSTIFF

For **CTYPE** = **BENDSTIFF** the format is:

CROSSECTION MYPIPENAME BENDSTIFF NBNOD NODEID ID OD RKS MATNAME

where:

NBNOD: Local node number in the bending stiffener definition.

NODEID: Corresponding global node ID in the bending stiffener definition.

ID: Inner diameter (unit: L).

OD: Outer diameter (unit: L).

RKS: Damping ratio (unit: -).

MATNAME: Material name

2.10.4 BENDSTIFF-BEND

Note: The difference between **BENDSTIFF** and **BENDSTIFF-BEND** is that by using **BENDSTIFF-BEND** the torsional and axial stiffness are not accounted for.

For **CTYPE** = **BENDSTIFF-BEND** the format is:

CROSSECTION MYPIPENAME BENDSTIFF-BEND NBNOD NODEID ID OD RKS MATNAME

where:

NBNOD: Local node number in the bending stiffener definition.
NODEID: Corresponding global node ID in the bending stiffener definition.
ID: Inner diameter (unit: L).
OD: Outer diameter (unit: L).
RKS: Damping ratio (unit: -).
MATNAME: Material name

2.10.5 NLBENDSTIFF

For **CTYPE** = **NLBENDSTIFF** the format is:

```
CROSSECTION MYPIPENAME  NLBENDSTIFF NBNOD NODEID ID OD NOTH
RKS MATIN MATOUT
```

where:

NBNOD: Local node number in the bending stiffener definition.
NODEID: Corresponding global node ID in the bending stiffener definition.
ID: Inner diameter (unit: L).
OD: Outer diameter (unit: L).
NOTH: Number of integration segments in thickness direction.
RKS: Damping ratio (unit: -).
MATIN: Material name for inside bending stiffener
MATOUT: Material name for outside bending stiffener

2.10.6 NLBENDSTIFF-BEND

Note: The difference between **NLBENDSTIFF** and **NLBENDSTIFF-BEND** is that by using **BENDSTIFF-BEND** the torsional and axial stiffness are not accounted for.

For **CTYPE** = **NLBENDSTIFF-BEND** the format is:

```
CROSSECTION MYPIPENAME  NLBENDSTIFF-BEND NBNOD NODEID ID OD
NOTH RKS MATIN MATOUT
```

where:

NBNOD: Local node number in the bending stiffener definition.

NODEID: Corresponding global node ID in the bending stiffener definition.

ID: Inner diameter (unit: L).

OD: Outer diameter (unit: L).

NOTH: Number of integration segments in thickness direction.

RKS: Damping ratio (unit: -).

MATIN: Material name for inside bending stiffener

MATOUT: Material name for outside bending stiffener

2.10.7 Fatigue Data

The fatigue data are stored on user specified files, one for each of the different steel layer types.

The file names are defined by the **CFATFL** character string specified as part of the cross-section data, see Section 2.10.1. Each input line specified below, assumes one help text line given first. The content is as specified below:

One line on the format:

Help text line
NUSMOD NFDPO1 R1 IGERB1 INTCO1 SCF1 NFDPO2 R2 IGERB2 INTCO2 SCF2 SIGUTS [SCFMAX SCFMBY SCFMBZ SCFDAX SCFDBY SCFDBZ]

where either 12 or 18 parameters are allowed for. Each parameter is defined as:

NUSMOD The number of failure modes to be considered.

- **= 1** Longitudinal failure mode (transverse cracks).
- **= 2** Both longitudinal (transverse cracks) and transverse (longitudinal cracks along pressure armour due to stresses in cross-section plane) failure modes .

NFDPO1 The number of points in the fatigue S-N diagram for longitudinal failure mode.

R1 The R-ratio defined as $\sigma_{min}/\sigma_{max}$ for the S-N-diagram for all values of **IGERB** except 21 and 22 where **R1** is the mean stress applied in the fatigue test.

IGERB1 Method for taking the mean stress into account for longitudinal failure mode.

- **= 0** No mean stress is taken into account. Stress range calculated considering longitudinal stress range for tensile armour, von Mises for pressure armour.

= 1 Goodman interpolation mean stress calculated as $\sigma_{xx}+\sigma_{yy}+\sigma_{zz}$ where σ_{yy} and σ_{zz} only applies for the pressure armour. Stress range calculated considering longitudinal stress range for tensile armour, von Mises for pressure armour.

= 2 : Gerber interpolation mean stress calculated as $\sigma_{xx}+\sigma_{yy}+\sigma_{zz}$ where σ_{yy} and σ_{zz} only applies for the pressure armour. Stress range calculated considering longitudinal stress range for tensile armour, von Mises for pressure armour.

= 3 Goodman interpolation mean stress calculated as
$\bar{\sigma} = \sqrt{\sigma_{xx}^2 + \sigma_{yy}^2 + \sigma_{zz}^2 - \sigma_{xx}\sigma_{yy} - \sigma_{zz}\sigma_{yy} - \sigma_{xx}\sigma_{zz} + 3\sigma_{xy}^2 + 3\sigma_{zy}^2 + 3\sigma_{xz}^2}$
For tensile armour only σ_{xx} applies. Stress range calculated considering longitudinal stress range for tensile armour, von Mises for pressure armour.

= 4 Gerber interpolation mean stress calculated as
$\bar{\sigma} = \sqrt{\sigma_{xx}^2 + \sigma_{yy}^2 + \sigma_{zz}^2 - \sigma_{xx}\sigma_{yy} - \sigma_{zz}\sigma_{yy} - \sigma_{xx}\sigma_{zz} + 3\sigma_{xy}^2 + 3\sigma_{zy}^2 + 3\sigma_{xz}^2}$
For tensile armour only σ_{xx} applies. Stress range calculated considering longitudinal stress range for tensile armour, von Mises for pressure armour.

= 5 **Only valid for pressure armour.** Goodman interpolation mean stress calculated as the mean principal stresses in two directions Principle stress ranges calculated for the same directions and mean stress correction respectively applied to the two components. In the transverse direction the material is assumed to start from yield stress. The two reported modes from LIFETIME will be different, 1= longitudinal mode, 2= transverse mode.
$\Delta\sigma = \max(\Delta\sigma_j)$
$\bar{\sigma} = \frac{1}{2}(\sigma_j^{\max} + \sigma_j^{\min})\ j = 1, 2$

= 6 **Only valid for pressure armour.** Gerber interpolation mean stress calculated as the mean principal stresses in two directions. Principle stress ranges calculated for the same directions and mean stress correction respectively applied to the two components. In the transverse direction the material is assumed to start from yield stress. The two reported modes from LIFETIME will be different, 1= longitudinal mode, 2= transverse mode.
$\Delta\sigma = \max(\Delta\sigma_j)$
$\bar{\sigma} = \frac{1}{2}(\sigma_j^{\max} + \sigma_j^{\min})\ j = 1, 2$

= 7 **Only valid for pressure armour.** Goodman interpolation mean stress calculated as the Von Mises stress. Principle stress ranges calculated for the same directions and mean stress correction respectively applied to the two components. In the transverse direction the material is assumed to start from yield stress. The two reported modes from LIFETIME will be different, 1= longitudinal mode, 2= transverse mode. $\Delta\sigma = \max(\Delta\sigma_j)$
$\bar{\sigma} = \sqrt{\sigma_{xx}^2 + \sigma_{yy}^2 + \sigma_{zz}^2 - \sigma_{xx}\sigma_{yy} - \sigma_{zz}\sigma_{yy} - \sigma_{xx}\sigma_{zz} + 3\sigma_{xy}^2 + 3\sigma_{zy}^2 + 3\sigma_{xz}^2}\ j =$

$1, 2$

= 8 **Only valid for pressure armour.** Gerber interpolation mean stress calculated as the Von Mises stresses. Principle stress ranges calculated for the same directions and mean stress correction respectively applied to the two components. In the transverse direction the material is assumed to start from yield stress. The two reported modes from LIFETIME will be different, 1= longitudinal mode, 2= transverse mode.
$\Delta\sigma = \max(\Delta\sigma_j)$
$\bar{\sigma} = \sqrt{\sigma_{xx}^2 + \sigma_{yy}^2 + \sigma_{zz}^2 - \sigma_{xx}\sigma_{yy} - \sigma_{zz}\sigma_{yy} - \sigma_{xx}\sigma_{zz} + 3\sigma_{xy}^2 + 3\sigma_{zy}^2 + 3\sigma_{xz}^2}$ $j = 1, 2$

= 21 Assuming the fatigue curve given at constant mean stress. A Goodman interpolation of mean stress is applied as
$\bar{\sigma} = \sqrt{\sigma_{xx}^2 + \sigma_{yy}^2 + \sigma_{zz}^2 - \sigma_{xx}\sigma_{yy} - \sigma_{zz}\sigma_{yy} - \sigma_{xx}\sigma_{zz} + 3\sigma_{xy}^2 + 3\sigma_{zy}^2 + 3\sigma_{xz}^2}$
For tensile armour only σ_{xx} applies. Stress range calculated considering longitudinal stress range for tensile armour, von Mises for pressure armour.

= 22 Assuming the fatigue curve given at constant mean stress. A Gerber interpolation of mean stress is applied as
$\bar{\sigma} = \sqrt{\sigma_{xx}^2 + \sigma_{yy}^2 + \sigma_{zz}^2 - \sigma_{xx}\sigma_{yy} - \sigma_{zz}\sigma_{yy} - \sigma_{xx}\sigma_{zz} + 3\sigma_{xy}^2 + 3\sigma_{zy}^2 + 3\sigma_{xz}^2}$
For tensile armour only σ_{xx} applies. Stress range calculated considering longitudinal stress range for tensile armour, von Mises for pressure armour.

INTCO1 Method for interpolation in S-N diagram:

= 1 Both stress and N in log scale.

= 2 Stress in linear scale, N in log scale.

SCF1 Stress concentration factor longitudinal failure mode.

NFDPO2 The number of points in the fatigue S-N diagram for transverse failure mode.

R2 The R-ratio defined as min/ max for the S-N-diagram for all values of **IGERB** except 21 and 22 where **R1** is the mean stress applied in the fatigue test.

IGERB2 Method for taking the mean stress into account for transverse failure mode as defined for **IGERB2** above.

INTCO2 Method for interpolation in S-N diagram:

= 1 Both stress and N in log scale.

= 2 Stress in linear scale, N in log scale.

SCF2 Stress concentration factor transverse failure mode.

SIGUTS Ultimate stress (unit: MPa).

SCFMAX Stress concentration factor to be used for the mean axial stress (unit: -).

SCFMBY Stress concentration factor to be used for the mean normal curvature stress (unit: -).

SCFMBZ Stress concentration factor to be used for the mean transverse curvature stress (unit: -).

SCFDAX Stress concentration factor to be used for the dynamic axial stress (unit: -).

SCFDBY Stress concentration factor to be used for the dynamic normal curvature stress (unit: -).

SCFDBZ Stress concentration factor to be used for the dynamic transverse curvature stress (unit: -).

By specifying the last 6 parameters the SCF1 and SCF2 parameters will be overruled.
Then a number of NFDPO1 lines on the format:

POINT SRANGE NCYFAL

where

POINT The point number $POINT = 1, .., NFDPO1$.

SRANGE Stress range (in increasing order), starting with the minimum threshold value (unit: MPa).

NCYFAL Corresponding number of cycles to failure.

Then a number of NFDPO2 lines on the format:

POINT SRANGE NCYFAL

where

POINT The point number $POINT = 1, .., NFDPO2$.

SRANGE Stress range (in increasing order), starting with the minimum threshold value (unit: MPa).

NCYFAL Corresponding number of cycles to failure.

Comment:
Only numeric values in the fatigue data file.

No lines starting with # is possible in the fatigue data file.

One line with comment is possible in front of each data type.

EXAMPLE:

```
NUSMOD NFDPO1 R1  IBERGB1 INTCO1 SCR1  NFDPO2 R2  IBERGB2 INTCO2 SCR2 SIGUTS
2       2     400    22      1    1.0    2    400    22      1    1.0   1030
POINT STRESS         N-cycles
1     8.572553977    1.00E+10
2     1170.645142    7.00E+04
POINT STRESS N-cycles
1     8.572553977    1.00E+10
2     1170.645142    7.00E+04
```

2.11 DYNCONT - Control parameters for dynamic analysis

The parameters controlling the dynamic analysis are defined by the following format:

DYNCONT MSTAT ALPHA1 ALPHA2 ALPHA [BETA GAMMA]

where:

MSTAT: Parameter for choosing mass matrix type:

- **1 :** Concentrated mass matrix.
- **2 :** Consistent mass matrix.

ALPHA1: Mass proportional damping factor, α_1.

ALPHA2: Stiffness proportional damping factor, α_2.

ALPHA: α in the HHT-α time integration method.

BETA: β in the HHT-α time integration method.

GAMMA: γ in the HHT-α time integration method.

BETA and **GAMMA** are optional. If only **ALPHA** is specified, default values for **BETA** and **GAMMA** are used. The default values are $\gamma = \frac{1}{2}(1-2\alpha)$ and $\beta = \frac{1}{2}(1-\alpha)^2$. The recommended value for the α-parameter is $-1/3 \leq \alpha \leq 0$. By setting $\alpha = 0$, $\gamma = 1/2$ and $\beta = 1/4$, the Newmark β method with constant average acceleration is obtained.

EXAMPLE:

```
#         imass  alfa1  alfa2   alfa
DYNCONT   1      0.0    0.095   -0.05
```

2.12 DYNRES_ - dynamic results

By the **DYNRES_** cards, user selected time histories will be stored on the .dyn file on a format that enables plotting in XPOST and post-processing by DYNPOSTB. The results are available for all time steps. The following format is applied:

```
DYNRES_N, E OR I ...
```

where **N** means nodal results, **E** means element results and **I** means integration station results. Note that there exists a memory limitation for **DYNRES_** for memory demanding analyses using the restart option. The allowable options are defined below:

2.12.1 DYNRES_N

For **DYNRES_** with the **N** (nodal results) option the format is as follows:

```
DYNRES_N TYPE NODEID DOF [NODEID2 DOF2]
```

where

TYPE: Type of result, where allowable values are:

- **1 :** nodal displacement
- **2 :** nodal velocity
- **3 :** nodal acceleration
- **4 :** relative displacement

NODEID: Node ID.

DOF: Degree of freedom number:

- **1 :** global x-displacement
- **2 :** global y-displacement
- **3 :** global z-displacement
- **4 :** global x-rotation
- **5 :** global y-rotation
- **6 :** global z-rotation

NODEID2: ID of node 2 (optional, applies only for **TYPE**=**4**).

DOF2: DOF of node 2 (optional, applies only for **TYPE**=**4**).

If the optional numbers **NODEID2** and **DOF2** are specified the relative displacements between the two nodes will be presented.

2.12.2 DYNRES_E

For elements of type PIPE and CONT, **DYNRES_** with the **E** option (element results) can be applied. The format is as follows:

```
DYNRES_E TYPE ELID ELNOD DOF [REFSYS]
```

where:

TYPE: Type of result where allowable values are:

- **1 :** Displacements.
- **2 :** Internal forces and moments.

ELID: Element ID.

ELNOD: Element node (max. 2 for PIPE and SPRING, max. 1 for CONT and BODY502)

DOF: Degree of freedom number. The meaning is element type dependent:

PIPE31-33, SPRING137, BODY502 : element types with **TYPE**=**1**,**2**

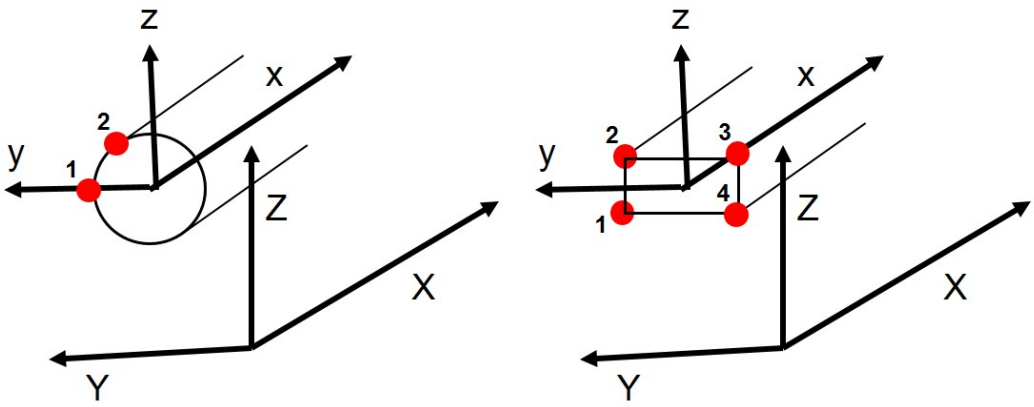
- **1 :** x-displacement/x-force
- **2 :** y-displacement/y-force
- **3 :** z-displacement/z-force
- **4 :** x-rotation/x-moment
- **5 :** y-rotation/y-moment
- **6 :** z-rotation/z-moment

CONT : element types with **TYPE**=**1**,**2**

- **1 :** x-displacement/x-force
- **2 :** y-displacement/y-force
- **3 :** z-displacement/z-force
- **4 :** x-rotation/x-moment (only for CONT126)

REFSYS: Reference coordinate system for the result, which can be assigned the values **LOCAL** or **GLOBAL** where the former is default. **REFSYS** is only implemented for **TYPE**=**2** and the CONT164 element, i.e. **LOCAL** will be applied for all other element types. Optional parameter.

For the CONT element types, the z-axis of the local system is pointing outwards from the master geometry in the contact point normal direction and the tangential plane is spanned by the x- and y-axes. Further, the output contact force acts on the master geometry for **REFSYS**=**LOCAL**, while it acts on the slave geometry for **REFSYS**=**GLOBAL**. The local system for the SPRING and PIPE element types coincide with the element system as defined by the **ELORIENT** card in Section 2.19.



(a) Tubular IP points (b) Rectangle IP points

Figure 2.17: Cross section integration point (IP) definition

2.12.3 DYNRES_I

For elements types PIPE33, PIPE52 (tensile armour), HSHEAR352 (tensile armour), HSHEAR353 and HSHEAR364 (tensile armour and manually modelled helices) , **DYNRES_** with the **I** option (integration station results) can be applied. For definition of integration points for tubular and rectangular csoss sections, see Fig. 2.17. The format is as follows:

DYNRES_I TYPE ELID IGAU IPOINT [IPOINT2]

TYPE: Type of result, where allowable values are:

- **1 :** sigma-xx (unit: FL^{-2})
- **2 :** strain-xx (unit: -)

ELID: Element ID.

IGAU: Element gauss station 1-3, 1 and 3 are at element ends, see Fig. 2.18 for illustration.

IPOINT: Integration point number, max. **NPOINT** see Section 2.6 and Fig. 2.18.

IPOINT2: Conceptual integration point. This applies to tensile armour models applying either PIPE52 or HSHEAR364 which do not have a visual model of the local cross-section. It is then possible to decide where in the cross-section profile (rectangular or circular) which is located at integration point IPOINT in the overall pipe cross-section the stress is to be calculated. The maximum value is 4. When moving along the positive helix axis: For rectangle 1= inner left, 2=upper left, 3= upper right and 4=lower right. For a tubular section: 1=left, 2 =outer surface, 3= right, 4=inner surface

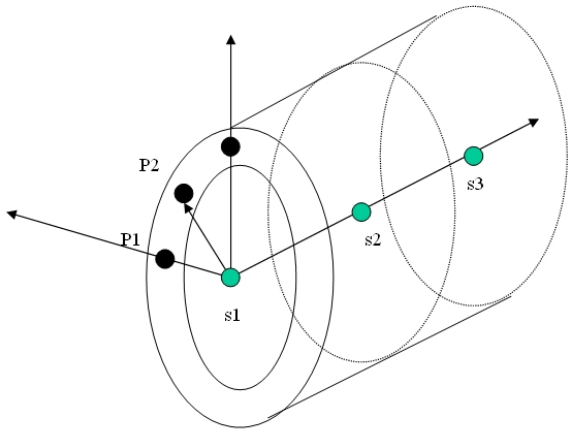


Figure 2.18: Element Gaussian station numbers s1 to s3 along the element axis. Integration points P1 and P2.

EXAMPLE:

```
#           type  elno end dof refsysy
DYNRES_E    2       2    1   1
DYNRES_E    2     440    1   1 global
DYNRES_E    2     439    1   1 global
#           type node  dof
DYNRES_N    1    3001    1
DYNRES_N    2    3001    1
DYNRES_N    1    3001    2
DYNRES_N    2    3001    2
```

2.13 ELCON - Element connectivity and properties

The elements in BFLEX2010 are organised into element groups each having a specific name. Each group is further defined by a reference to element type and material type. By the **ELCON** command, the user defines the element group, references to element and material types as well as the element connectivity. An overview of element types, associated allowed material types and number of element nodes for each element type is found in Table 2.1. Note that for some contact elements, only the master node ID of the element connectivity matrix is given. The following format is applied:

ELCON ELGR ELTY MATNAME ELID NOD1 [NOD2 NOD3 NOD4]
[REPEAT N NELINC NODINC]

where:

ELGR: Element group name.

ELTY: Element type. The complete list of available element types is given in Table 2.1 below.

MATNAME: For the contact element CONT126: Name of contact surface. See the **COSURFPR** command in Section 2.8. For BODY502 the value must be **NONE**. For PIPE52, HSHEAR352, HSHEAR353, HSHEAR363, HSHEAR364, HCONT453, HCONT463, HCONT464 and HCONT454: Name of cross section, see commands FLEXCROSS or 353FLEXCROSS in Section 2.10. With the exception of PIPE52 and HSHEAR352, it is also allowed to specify a material name instead to allow the user to model any type of layered helical structure. Then the cross-section properties are established by means of the material and geometry specified via the **SHEARHELIX** or **SHEAR2HELIX** options, see Section 2.20 for element types HSHEAR353, HSHEAR363 and HSHEAR364. It is to be noted that by introducing a userdefied material for HCONT453 or HCONT463 it is possible to model a bonded pipe, by simply introducing the word **USERDEFINED** at the end of the CONTACT material specification, see Section 2.26. For all other element types: Material name. Note that HSHEAR353 and HSHEAR352 assumes that the stress-free configuration is defined along the global x-axis and in the global x-z plane

ELID: First element ID number

NOD1: Node ID

NOD2: Node ID, optional depending of element type

NOD3: Node ID, optional depending of element type

NOD4: Node ID, optional depending of element type

If the **REPEAT** command is introduced, the previous sequence is repeated:

N: Number of repeats.

NELINC: Element increment.

NODINC: Node increment.

The element types that are allowed for in the version 3.3.1 of the program are shown in Table 2.1. The material is described in the **MATERIAL** card and the element type is the **ELPROP** card.

Table 2.1: Overview of BFLEX2010 element library.

Type	Name	Description	Element property type	Material type	Nodes	Elforce system	Elforce - Confor/ Stress units
31	PIPE31	3D beam constant axial strain and torsion	PIPE	LINEAR	2	Local	F,FL,FL^{-2}
33	PIPE33	3D beam constant axial strain and torsion	PIPE	ELASTOPLASTIC	2	Local	F,FL,FL^{-2}
52	PIPE52	3D flexible pipe, moment based (ITCODE=21)	See Section 2.10	LINEAR ELASTIC	2	Local	F,FL,FL^{-2}
42	COMPIPE42	3D beam constant axial strain and torsion	COMPIPE	RESULTANT	2	Local	F,FL,FL^{-2}
111	CABLE111	3D cable constant axial strain	CABLE	LINEAR EPCURVE HYCURVE	2	Global	F,FL^{-2}
126	CONT126	3D Seabed contact element (x and y force relative to last contact)	SOILCONTACT	n-dir: CONTACT s/t-dir: EPCURVE, HYCRUVE	1	Local	F,FL^{-1}
130	CONT130	3D Bellmouth contact element	BELLMOUTH	n-dir: CONTACT s/t-dir: EPCURVE, HYCURVE	1(3)	Local	F
137	SPRING137	3D non-linear spring element	GENSPRING	GENSPRING	2	Local	F,FL
152	CONT152	3D Pipe in body element	BELLMOUTH	n-dir: CONTACT s/t-dir: EPCURVE, HYCURVE	1(2)	Local	
150	SEA150	Sea element		SEA	4		
164	CONT164	3D Stinger contact element	ROLLER	n-dir: CONTACT ISOCONTACT ISOKXYCONTACT	1(3)	Local	F
231	HELIX231	3D elastic helix element	HELIX	structural: ELASTIC n-dir: CONTACT, - EPCURVE/ HYCURVE	2(3)	Local	F,FL,FL^{-2}
233	HELIX233	3D elastoplastic helix element	HELIX	structural: ELASTOPLASTIC n-dir: CONTACT, - EPCURVE/ HYCURVE	2(3)	Local	F,FL,FL^{-2}
234	HELIX234	3D elastic Sz element	SZHELIX	structural: ELASTIC n-dir: CONTACT, - EPCURVE/ HYCURVE	2(3)	Local	F,FL,FL^{-2}
235	HELIX235	3D resultant Sz element	SZHELIX	structural: RESULTANT	2(3)	Local	F,FL,FL^{-2}

Continued on next page

Type	Name	Description	Element property type	Material type	Nodes	Elforce system	Elforce - Confor/ Stress units
352	HSHEAR352	3D flexible pipe, sandwich based element (IT-CODE=0)	See Section 2.10	LINEAR ELASTIC	4	Local	F,FL,FL^{-2}
353	HSHEAR353	3D flexible pipe, sandwich based element	SHEARHELIX or by Section 2.10	LINEAR ELASTIC	4	Local	F,FL,FL^{-2}
363	HSHEAR363	3D flexible pipe, beam/thinshell element	SHEARHELIX, SHEAR2HELIX or by Section 2.10	LINEAR ELASTIC	3	Local	F,FL,FL^{-2}
364	HSHEAR364	3D flexible pipe, beam/thickshell element	SHEARHELIX, SHEAR2HELIX or by Section 2.10	LINEAR ELASTIC	4	Local	F,FL,FL^{-2}
437	HELSPR437	3D helix spring element	HELSRPING	HELSPRING EPCURVE/ HYCURVE	2	Local	F,FL
453	HCONT453	3D Armour-armour normal contact element	LAYERCONTACT or by Section 2.10	n-dir: CONTACT ISOCONTACT s/t-dir: EPCURVE, HYCURVE FRICONTACT	4	Local	FL^{-1}
454	HCONT454	3D Armour-armour lateral contact element	LAYERCONTACT or by Section 2.10	n-dir: CONTACT ISOCONTACT s/t-dir: EPCURVE, HYCURVE FRICONTACT	6	Local	FL^{-1}
463	HCONT463	3D Armour-sheath contact element	LAYERCONTACT or by Section 2.10	n-dir: CONTACT ISOCONTACT s/t-dir: EPCURVE, HYCURVE FRICONTACT FRICVISC	3	Local	FL^{-1}
464	HCONT464	3D Layer normal contact element	LAYERCONTACT or by Section 2.10	FRICONTACT FRICVISC	2	Local	F L^{-1}
473	HCONT473	3D Helix-sheath contact element	LAYERCONTACT	n-dir: CONTACT ISOCONTACT s/t-dir: EPCURVE, HYCURVE FRICONTACT	7	Local	FL^{-1}
502	BODY502	3D Body element	BODY	NONE	1	Local	

2.13.1 PIPE

Note: It is noted that PIPE52 is used for flexibles pipes and bend stiffeners only.

PIPE31-33 are 2-noded beam elements, differing with respect to material model. In order to consistently handle the pressure effect, it is necessary to work on element level. Hence the beam elements in BFLEX2010 are basically PIPE elements assuming

thin walled tubular cross-section with constant radius and thickness along each element. Accordingly, the concept 'PIPE' has been used rather than 'BEAM'. As a consequence of this, the pipe geometry properties are linked to the element concept, basically assuming that all PIPE elements in an element group have the same properties.

However, overruling this concept is allowed for by using the **NODPROP** data card, see Section 2.29. The **NODPROP** data card is made to ease modelling of bellmouth/bend stiffener geometries.

2.13.2 COMPIPE

COMPIPE42 is a pipe element based on resultant material curves that may be either hyperelastic or elastoplastic. This is to allow modelling of e.g. bending restrictors or flexible pipes/umbilicals.

PIPE52 is a 2 noded 12 DOF beam element used to model flexible pipes. This includes the tensile armours, the remaining layers (the core) and the bend stiffener. To model a flexible pipe, separate element groups need to be defined for each armour layer and the core. The mechanical properties of each layer is established from the **CROSSECTION** input, see Section 2.10.

2.13.3 CABLE

CABLE111 is a 3D 6 DOF cable element allowing elastic and non-linear elastic stiffness to be described.

2.13.4 CONT

CONT164 is a 3D 3-noded roller element which basically assumes that contact is obtained between a user defined cylinder attached to node 1 and an arbitrary position between two pipe or cable nodes (node 2 and 3).

CONT130 is a pipe in pipe element to model bellmouth contact, which basically assume that contact is obtained between a user defined cylinder attached to node 1 and an arbitrary position between two pipe nodes (node 2 and 3). Since the contact search may become time consuming, constraints may be introduced in the contact search by the **CONTINT** card, see Section 2.5. Note that only the master node is included in the connectivity input, as the other ones are set by the program. CONT130 requires that there are two node and element systems, one for the pipe node and one for the other bellmouth/pipe structures. The master node will be the pipe node of the master node

system which may come in contact with two slave bellmouth structure/pipe nodes of the second node system.

CONT126 is a 1-noded contact element which is linked to the contact surface definition, see Section 2.8. The element includes the torsion coupling effect. Since the contact search may become time consuming constraints may be introduced in the contact search by the **CONTINT** card, see Section 2.5.

CONT152 is a two noded contact element to be applied for contact between pipe and body element. One node per element (the body node) is specified in the connectivity input. The second node is a node on the pipe. The pipe nodes beeing closest to the body object in the initial configuration is selected. Note that this element is linked to the geometrical object linked to the body element CONT152. and if an eccentricity is given for the body element the same eccentricity must be given for the contact element in order to correctly capture the same eccentricity and associated bending moments.

2.13.5 SPRING

SPRING137 is a 2-noded spring elemens where it is possible to attach mass, damping and arbitrary material curves for 6 nodal DOFs.

2.13.6 HELIX

HELIX 231/233 are 3D helix elements which may be attached to a centre group of elements by either sharing the same nodal DOFs or by using a combination of HELSPR437 to simulate axial interaction and constraint coefficients, see Section 2.4. The same governs for the element HELIX234/235 which is made to allow modelling of both helices and Sz-helices.The elements are from a user point of view 2-noded, i.e. they work as pipe elements with respect to node connectivity. There is one additional internal node (which is handled internally by the program) that includes the radial and transverse DOF allowing the user to simulate radial gap effects. This requires additional element data with respect to radial interaction material curves. It is noted that these elements assumes that the helix can be represented by a space curve wound around a cylinder with constant radius. Axisymmetric loads, which means loads where the cylinder shape of the supporting structure is kept, includes axial force, pressure and torsion. The structural response due to these loads is assumed to be uncoupled from bending effects, where a separate model is applied. The bending model uses the radial pressure as a result of the HELIX element property definition, which defines the link to special material data. Hence by using a combination of PIPE and HELIX elements no friction effects due to axisymmetric loads is included in the model.

2.13.7 HSHEAR

HSHEAR352 is a 4 noded 16 DOF curved sandwich beam element dedicated to the modelling of helices. It includes 12 beam DOFs corresponding to those of PIPE31 plus 4 helical DOFS along the helix, allowing longitudinal slip to be modelled.

HSHEAR353 is a 4 noded 26 DOF curved beam element dedicated to the modelling of helices. It includes 12 beam DOFs corresponding to those of PIPE31 plus 6 helical DOFS at each end of the corresponding helix. In addition 2 two internal DOFs are used to allow accurate description of the longitudinal slip process. The torsion DOFs 4 and 10 are dummy. However, if the model includes a thick-walled based modelling procedure by applying HSHEAR364 and HCONT464, then these DOFs are applied to model the through thickness strain of the helix layer.

HSHEAR363 is a 3 noded 15 DOF beam-thinshell element dedicated to the modelling of the pressure armour, the anti-buckling tape and the plastic layers. It consists of 2 beam nodes with 6 DOFs each corresponding to those of PIPE31, plus one node with 3 DOFs to describe radial motion. As of today, only DOF 1 for the 3-DOF node is active, which means that DOFs 2 and 3 must be fixed. The 3-DOF node has no geometric meaning and can be arbitrary positioned, because it is only used to carry the radial motion in the local element system. The thin shell concept in this context means that the through thickness strains are ommitted, i.e. only axial and hoop strains are included.

HSHEAR364 is a 4 noded 18 DOF beam-thickshell element dedicated to the modelling of the pressure armour, the anti-buckling tape and the plastic layers. It consists of 2 beam nodes with 6 DOFs each corresponding to those of PIPE31, plus two nodes with 3 DOFs associated to each side of the thickness direction, describing radial motion and through thickess strain. As of today, only DOF 1 for the 3-DOF nodes is active, which means that DOFs 2 and 3 must be fixed. The 3-DOF nodes have no geometric meaning and can be arbitrary positioned, because they are only used to carry the radial motion in the local element system. The thick shell concept in this context means that constant through thickness strains are included for each element in addition to axial and hoop strains, when appropriate. For helix layers, the circumferential strain is given by the local layer pressure (2D or 3D Hooke's law). Note that it is not allowed to combine HSHEAR363 and HSHEAR364 elements.

Cross-sections that consist of concentric helix and tubular layers can be modelled by combining HSHEAR364 and HCONT464. By specifiying a non-zero static and dynamic friction coefficient in the material model for HCONT464, friction moments arising from the helix contact interfaces can be included, see Section 2.26.12.

2.13.8 HCONT

HCONT453, HCONT454, HCONT463, HCONT464 and HCONT473 are respectively 4 noded, 6 noded, 3 noded, 2 noded and 7 noded hybrid mixed contact elements used to describe contact between other HSHEAR elements. HCONT453 models contact between two helix layers of HSHEAR353 elements, HCONT454 deals with contact between helices in the same layer with HSHEAR353 elements and HCONT463 models contact between the HSHEAR363 elements and the HSHEAR353 elements. HCONT464 is a 2 noded contact element to carry the contact load between layers of HSHEAR364 or HSHEAR364 elements (linking the 3-DOF nodes). This allows for including through thickess deformation in flexible pipes and other layered helix structures. The contact element **HCONT473** considers contact between **HSHEAR353** and **HSHEAR363** or **HSHEAR364**, however, also including their respective centreline axial and torsion motions in the relative displacement and friction. The element allows for arbitrary large displacement sliding (The other ones assume small displacemenets). This means that sliding of one helix relative to the remaining cross-section due to temperature, curvature and gravity can be assessed in addition to evaluating local bending stresses at the end fitting. Also the significant hysteresis observed for torsion motion and its coupling to the bending motion can be descr

2.13.9 HELSPR

In order to investigate phenomena related to whether a helix will slide relative to the centre and carry gravity loads by itself, the HELSPR437 contact spring element is needed. This is a 2 noded element with only one DOF, i.e the axial DOF along the helix curvilinear system. For each node of the HELSPR437 element, the radial pressure from all helix elements attached to node 2 of the element are used to scale the spring friction properties (force= fric. coefficient*total radial force).

2.13.10 SEA

SEA150 is an element that simulates the sea properties. In BFLEX2010, all physical effects is forced into the concept of elements and nodes, and so also the sea which is represented by an arbitrary set of 4-noded shell elements. The application of buoyancy force effects requires that a contact interface is defined between the structural element and the sea element groups, see Section 2.5.

2.13.11 BODY

BODY502 is a one node element, that can be connected to a geometry, see Section 2.23.

EXAMPLE:

```
# Elcon  input:
# Pipe section
#-----------------------------------------
#      group     elty       material  elID   n1  n2
ELCON  ormpipe1  pipe31     pipemat1     1    1   2
#         n j k
REPEAT  340 1 1
#
#      group     elty       material  elID   n1   n2
ELCON  ormpipe2  pipe31     pipemat1   341  341  342
#         n j k
REPEAT  100 1 1
#
#      group     elty       name of surface   elID   n1
ELCON  seabed    cont126    cosurf1           1001    1
#         n j k
REPEAT  200 1 1
```

2.14 ELDAMP - Element damping properties

Element can be specified for PIPE, SPRING and CONT elements. Damping constants c_i are given for each DOF i. The unit depends on element type, in which the damping constants for PIPE and soil CONT elements are per unit length (units: $\mathrm{FL^{-1}T^{-1}}$ for translation and $\mathrm{FT^{-1}}$ for rotation), while lumped constants are applied for SPRING, CONT124, 164, 130 and 170 (units: $\mathrm{FT^{-1}}$ and $\mathrm{FLT^{-1}}$). For two-noded elements with physical length, the damping load relation for DOF i at element nodes A and B reads:

$$\begin{bmatrix} S_{iA} \\ S_{iB} \end{bmatrix} \begin{bmatrix} c_i\frac{1}{2} & -c_i\frac{1}{2} \\ -c_i\frac{1}{2} & c_i\frac{1}{2} \end{bmatrix} \begin{bmatrix} \dot{v}_{iA} \\ \dot{v}_{iB} \end{bmatrix}$$

In addition, a Rayleigh damping matrix which is proportional to the material stiffness matrix and the mass matrix can be given. The **ELDAMP** input data is only used for dynamic analyses.

The following format is applied:

```
ELDAMP GRNAME TYPE ...
```

where

GRNAME: Element group name.

TYPE: Element property type which may have three options:

BEAM : Covers PIPE, SPRING and BODY502 element types.

CONTACT : Covers the CONT element types and only translation DOFs are involved.

RALEIGH : Can be applied for structural elements only, i.e. CABLE or BEAM elements. Note that CABLE elements only allow for Rayleigh damping.

The damping constants for the BEAM and RALEIGH options refer to the element coordinates, which may be eccentric relative to the element nodes if the **ELECC** card in Section 2.15 is applied. For the CONTACT option, the damping force acts at the contact point and is present only when the surfaces are in contact. The damping constants refer to the contact point coordinate system where the z-direction always coincides with the normal direction of the contacting surfaces.

2.14.1 BEAM

For **TYPE** = **BEAM** the following format applies:

```
ELDAMP GRNAME BEAM C1 C2 C3 C4 C5 C6
```

where

C1: Damping constant in element x-direction.

C2: Damping constant in element y-direction.

C3: Damping constant in element z-direction.

C4: Rotational damping constant about element x-axis

C5: Rotational damping constant about element y-axis

C6: Rotational damping constant about element z-axis

The damping constants **C1-C6** are set at analysis start-up and cannot be changed during analysis restarts.

2.14.2 CONTACT

For **TYPE** = **CONTACT** the following format applies:

```
ELDAMP GRNAME CONTACT C1 C2 C3
```

where

C1: Contact point damping constant in x-direction tangential to contact interface.

C2: Contact point damping constant in y-direction tangential to contact interface.

C3: Contact point damping constant in z-direction normal to contact interface.

The damping constants **C1-C3** can be changed by the user during analysis restarts.

2.14.3 RALEIGH

The following format is applied for **TYPE** = **RALEIGH**:

```
ELDAMP GRNAME RALEIGH ALFA1 ALFA2 [UPDATE=VALUE]
```

where

ALFA1: Rayleigh damping factor for mass matrix contribution.

ALFA2: Rayleigh damping factor for material stiffness matrix contribution.

UPDATE=VALUE: **UPDATE=** is a character string and **VALUE** is an integer equal to either **0** or **1**. For **VALUE=0** which is default, the initial material stiffness matrix is applied, and for **VALUE=1** the updated stiffness matrix is applied. The option is only available for PIPE33 and COMPIPE42 elements.

The damping factors **ALFA1** and **ALFA2** are set at analysis start-up for the option **UPDATE=0** and can then not be changed during analysis restarts. **ALFA1** and **ALFA2** can however be changed if **UPDATE=1** is applied. For two-noded elements, the same damping factors are applied at both element ends. Note that **ALFA1** and **ALFA2** are not dimensionless.

EXAMPLE:

```
#       grname      type     c1       c2        c3        c4       c5      c6
ELDAMP ormcontact contact 0.000    0.000     0.1010
ELDAMP ormpipe2   beam    0.000e0 0.0000e0 0.0000e0 0.000e0 0.00e0 0.000e0
#       grname      type      alfa1_mass alfa2_stiff
ELDAMP ormpipe2   raleigh      0.000e0   0.01000e0
```

2.15 ELECC - Element eccentricity

The **ELECC** command allows the user to define element eccentricities relative to an arbitrary element node. The general format is as follows:

```
ELECC ELTYP ELID ELEND ...
[REPEAT  ...]
```

where

ELTYP: Type of element, which may have the following values:

BEAM : allowed element types PIPE31-33,52, COMPIPE42, BODY502, CONT152 and SPRING137.

RADIUS : allowed element type CONT164.

STINGER : allowed element types CONT164.

2.15.1 BEAM

The **ELECC** card format has the following format for the **BEAM** option :

```
ELECC BEAM ELID ELEND XECC YECC ZECC
[REPEAT N ELINC]
```

where

ELID: Element ID number.

ELEND: Element end number, values **1** and **2** allowed.

XECC: Eccentricity in x-direction, see Fig. 2.2.

YECC: Eccentricity in y-direction.

ZECC: Eccentricity in z-direction.

The **REPEAT** card has the following format for the **BEAM** option:

N: Number of repeats.

ELINC: Element increment.

For PIPE 31-33 , the element origin eccentricity is defined in global coordinates. The eccentricity vector rotates together with the element system which follows the rotation of the nodes, including a possible initial rotation defined by the **NOORIENT** card. The nodal rotations contains both rigid body and deformational rotations, meaning that the element eccentricity refers to the deformed pipe configuration.

The eccentricity is defined in global coordinates for CONT152, BODY502 and SPRING137. The eccentricity vector rotates together with the element system which follows the rotation of the element node specified as **NOD1** in the **ELCON** card, see Section 2.13, including a possible initial rotation defined by the **NOORIENT** card.

EXAMPLE:

```
#     type    elid   end  ex     ey  ez
ELECC beam    3000   1    71.56  0   25.000
```

2.15.2 STINGER

The format of the **ELECC** card is as follows for the **STINGER** option:

ELECC STINGER ELID ELEND XECC YECC ZECC YPHI DX1 DY1 DZ1 DX2 DY2 DZ2 [REPEAT N ELINC DS DYPHI [RADN]]

where

ELID: Element ID number.

ELEND: Element end number. Must apply **ELEND**=1.

XECC: x-component eccentricity in element coordinates relative to the master node, see e_x in Fig. 2.19 (unit: L).

YECC: y-component eccentricity in element coordinates relative to the master node (unit: L).

ZECC: z-component eccentricity in element coordinates relative to the master node, see e_z in Fig. 2.19 (unit: L).

YPHI: Orientation angle of the roller system relative to the element system, see φ in Fig. 2.19. Defined as positive for rotation about the negative y-axis of the element coordinate system (unit: rad).

DX1: x-component eccentricity in roller coordinates for the first roller end relative to the point defined by **XECC**, **YECC** and **ZECC** (unit: L).

DY1: y-component eccentricity in roller coordinates for the first roller end relative to the point defined by **XECC**, **YECC** and **ZECC**, see dy1 in Fig. 2.19 (unit: L).

DZ1: z-component eccentricity in roller coordinates for the first roller end relative to the point defined by **XECC**, **YECC** and **ZECC** (unit: L).

DX2: x-component eccentricity in roller coordinates for the second roller end relative to the point defined by **XECC**, **YECC** and **ZECC** (unit: L).

DY2: y-component eccentricity in roller coordinates for the second roller end relative to the point defined by **XECC**, **YECC** and **ZECC** (unit: L).

DZ2: z-component eccentricity in roller coordinates for the second roller end relative to the point defined by **XECC**, **YECC** and **ZECC** (unit: L).

The eccentricities **XECC**, **YECC** and **ZECC** refer to the element coordinate system, see x and z in Fig. 2.19, which follows the rotation of the element node specified as **NOD1** in the **ELCON** card, see Section 2.13, including a possible initial rotation defined by the **NOORIENT** card. The remaining six eccentricities refer to the roller coordinate system, see x_1 and z_1 in Fig. 2.19. The roller coordinate system is obtained by rotating the element coordinates system about its negative y-axis by the magnitude **YPHI**.

The **REPEAT** card allows for modelling of stingers with constant or linearly changing radius of curvature. In order to utilize the **REPEAT** card for this purpose, the roller axis must be directed along the y_1-axis, i.e. **DX1**=**DX2**=**DZ1**=**DZ2**=0, which may require a specific orientation of the element or the master node by means of the **ELORIENT** and **NOORIENT** cards, respectively. The format of the **REPEAT** card is as follows for the **STINGER** option:

N: Number of repeats for defining rollers $i=\{2,3,...,N\}$.

ELINC: Element ID increment.

DS: Constant arch length increment between corresponding points on x_1-axes of rollers $i-1$ and i. **DS** is defined positive in direction of the negative x_1-coordinate axis of roller $i-1$ for positive **DYPHI**, see ds in Fig. 2.19. For negative **DYPHI**, the positive direction of **DS** is in the direction of the positive x_1-coordinate axis of roller $i-1$ (unit: L).

DYPHI: Constant angle increment of roller coordinate system orientation angle **YPHI**, see $d\varphi$ in Fig. 2.19. Defined as positive for rotation about the negative element y-axis. If the parameter **RADN** is given, **DYPHI** will be applied only for roller $i=2$ while the increment of **YPHI** for the remaining rollers will be set equal to **DS** divided by the linearly interpolated curvature radius (unit: rad).

RADN: Stinger radius of curvature between the two last rollers, $i=N-1$ and $i=N$. Linear interpolation of the curvature radius is then applied between rollers $i=2$ to $i=N-1$. Optional parameter. (unit: L).

If the optional parameter **RADN** is omitted, the y_1-axes of the rollers will be placed on a curve with constant radius of curvature in the xz-plane equal to **DS** divided by **DYPHI**. If the parameter **RADN** is given, the stinger radius of curvature will be equal to **DS** divided by **DYPHI** between rollers $i=1$ and $i=2$, and thereafter change linearly towards **RADN** between rollers $i=N-1$ and $i=N$.

The example below utilizes the **ELECC STINGER** option to create a spiral with radius of curvature increasing linearly from $\frac{ds}{d\varphi}=\frac{1.0}{0.05}=20.0$ to 2.0:

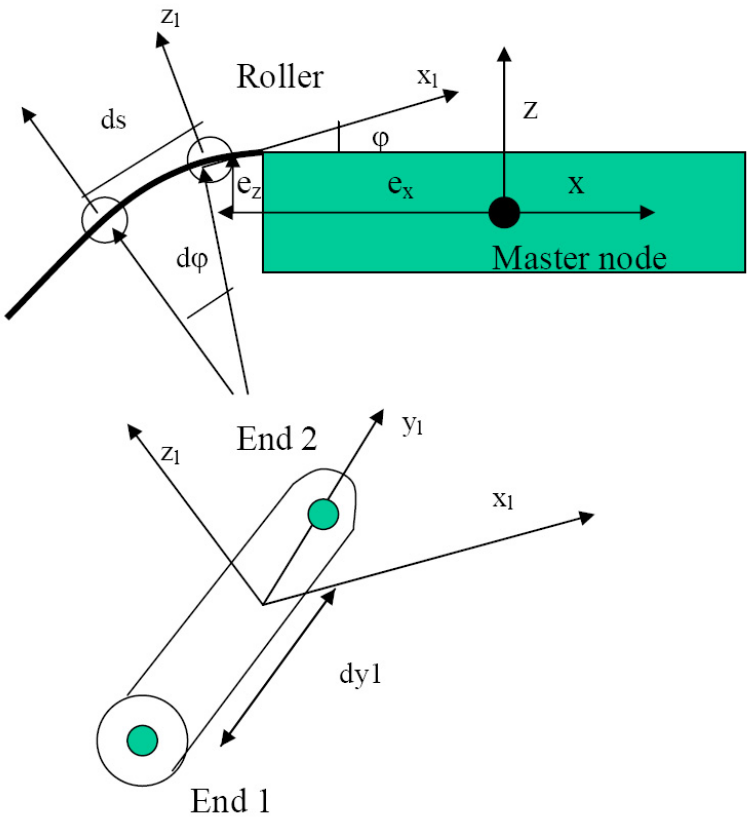


Figure 2.19: Element coordinates (x, y, z), roller coordinates (x_1, y_1, z_1), orientation angle φ of the roller coordinate system and incremental change of arch length ds and orientation angle $d\varphi$ between rollers.

EXAMPLE:

```
# Elecc data:
#     type    elid  elend  xecc  yecc  zecc  yphi  dx1   dy1  dz1  dx2  dy2  dz2
ELECC stinger  1      1     0.0   0.0   0.0   0.0   0.0  -1.0  0.0  0.0  1.0  0.0
#
#        n   elinc  ds   dyphi  radn
repeat  100    1    1.0   0.05   2.0
```

where the resulting geometry is shown in Fig. 2.20.

2.15.3 RADIUS

The **ELECC RADIUS** option is used for modelling of rotation-symmetric geometries consisting of several rollers. The same functionality is available also through the **REPEAT** card for **ELECC STINGER**, however, the **RADIUS** option has increased flexibility as the roller can rotation-symmetric about each of the three coordinate axes. The

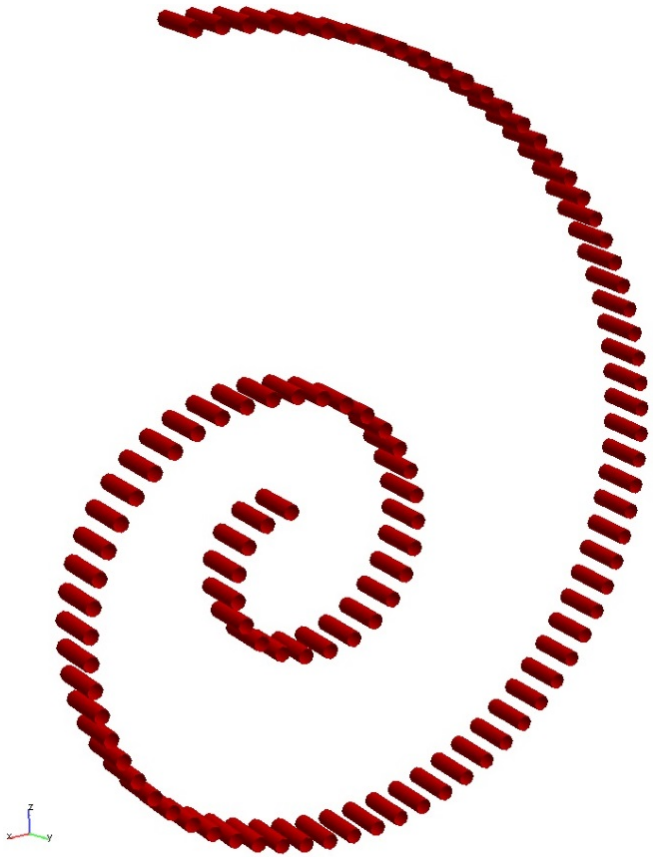


Figure 2.20: Modelling of spiral with linearly changing radius of curvature.

ELECC card has the following format for the **RADIUS** option:

```
ELECC RADIUS ELID ELEND IAXIS DX1 DY1 DZ1 DX2 DY2 DZ2
[REPEAT N ELINC DPHI]
```

where

ELID: Element ID number.

ELEND: Element end number. Must apply **ELEND**=1.

IAXIS: The element axis which the roller end eccentricity vectors is rotated about. The eccentricity vector for end 1 is defined by **DX1**, **DY1** and **DZ1** and by **DX2**, **DY2** and **DZ2** for end 2. The following values are allowed:

- **1 :** Rotation about element x-axis.
- **2 :** Rotation about element y-axis.
- **3 :** Rotation about element z-axis.

Dummy parameter if no **REPEAT** card is given or if only **N**=1 repeat is applied.

DX1: x-component eccentricity in element coordinates for the first roller end relative to the master node (unit: L).

DY1: y-component eccentricity in element coordinates for the first roller end relative to the master node (unit: L).

DZ1: z-component eccentricity in element coordinates for the first roller end relative to the master node (unit: L).

DX2: x-component eccentricity in element coordinates for the second roller end relative to the master node (unit: L).

DY2: y-component eccentricity in element coordinates for the second roller end relative to the master node (unit: L).

DZ2: z-component eccentricity in element coordinates for the second roller end relative to the master node (unit: L).

The eccentricities **DX1**, **DY1**, **DZ1**,**DX1**, **DY1** and **DZ1** refer to the element coordinate system, which follows the rotation of the element node specified as **NOD1** in the **ELCON** card, see Section 2.13, including a possible initial rotation defined by the **NOORIENT** card.

The **REPEAT** card has the following format for the **RADIUS** option:

N: Number of repeats.

ELINC: Element ID increment.

DPHI: Constant angle increment for rotation about **IAXIS**=1,2 or 3. Positive rotation directions are defined according to the right-hand rule (unit: rad).

Contact elements based on line-line search algoritms are known to fail for parallell lines. In contact modelling of two parallell pipelines, this issue can be avoided effectively by using the **RADIUS** option for modelling of the external pipe radius according to:

EXAMPLE:

```
# Elecc data
#     type    elid  elend  iaxis  dx1   dy1   dz1   dx2   dy2   dz2
ELECC radius  1       1       2   0.02  0.0  -0.3  -0.02  0.0  -0.3
#
#        n   elinc   dphi
repeat  49    1     -0.065
```

which defines a half circle consisting of 49 overlapping rollers as shown in Fig. 2.21. The roller with element ID 1 is located at $z = -0.3$ and spans from $x = 0.02$ to $x = -0.02$. The element IDs are increasing in the direction of negative rotation about the y-axis. All of the rollers are located approximately on the circle $x^2 + z^2 = 0.3^2$, and overlaps each other by half of the roller length to avoid deadzones in the contact search.

The example below defines 20 rollers as shown in Fig. 2.22 spanning from z = −4.0 to z = 4.0:

EXAMPLE:

```
# Elecc data
#     type    elid  elend  iaxis  dx1  dy1   dz1  dx2  dy2  dz2
ELECC radius  1       1       3   3.0  0.0  -4.0  3.0  0.0  4.0
#
#        n  elinc  dphi
repeat  20  1      0.2
```

where element ID 1 is located at coordinate x = 3.0 and y = 0.0. The other rollers are also located on the circle $x^2 + y^2 = 3^2$ with the element IDs increasing in the direction of positive rotation about z-axis.

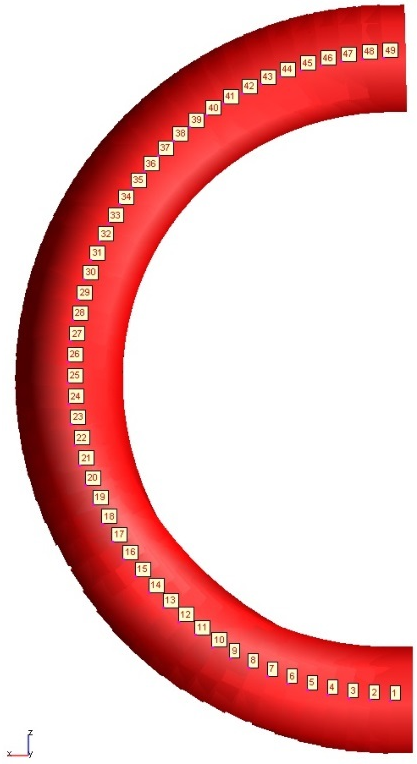


Figure 2.21: Rotation-symmetric geometry for pipe-pipe external contact interaction modelled by the **RADIUS** option.

2.16 ELHIST - element time histories

By the **ELHIST** command the user is allowed to manipulate the properties of elements by associating different element ID numbers to a load history, see Section 2.35. If the

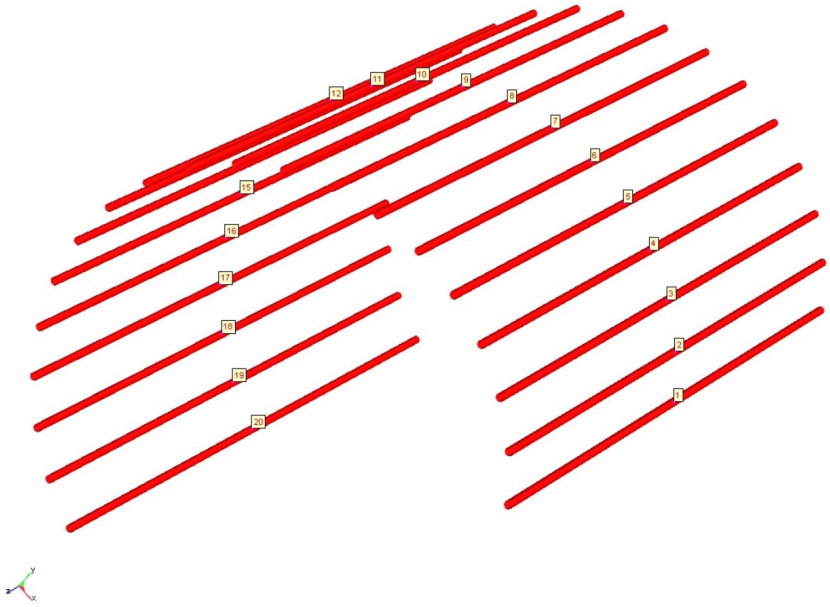


Figure 2.22: Rotation-symmetric geometry modelled by the **RADIUS** option.

value of the associated time history scaling factor is positive, the elastic properties will be scaled according to the factor given. The user is allowed to define an arbitary sequence of element deactivation (death) and element activation (birth) by sequently associating negative or positive numbers in the time history. As for now this feature only exists for element types PIPE31, HSHEAR353, HSHEAR363 and HSHEAR364. Please note that in the case of element death, the equation system will become singular, unless other active elements are connected to the nodes, loosing contributions from the dead elements. This feature must therefore be used with care.

Note that if a negative scaling factor is specified, the element will be deactivated, i.e. no contribution will be included for this element. This will save computing time. However, it is not allowed to start with a negative factor at time zero.

Note that it is only the element stiffness that will be scaled. The stresses are not relevant, but will still be reported based on the specified Youngs modulus.

The following format is applied:

```
ELHIST ELID11 ELID12 HISTNO1
.. .. ..
ELIDN1 ELIDN2 HISTNON
```

where:

ELID11: The first element ID of sequence 1.

ELID12: The last element ID of sequence 1.

HISTNO1: The time history ID number for sequence 1.

ELIDN1: The first element ID of sequence n.

ELIDN2: The last element ID of sequence n.

HISTNON: The time history ID number for sequence n.

EXAMPLE:

```
#            Element history feature
#
#        elid1 elid2  histno
ELHIST   1     100    200
         200   300    200
#
#           Element history by thist
#       no     time    fac
THIST   200    0       1.0
               5       1.0
               6      -1.0
               10     -1.0
               11     0.5
```

The above will cause the elements 1-100 and 200-300 to have an elastic stiffness according to the nominal input given until 6 s. Then from 6 s and up to 11 s they will be removed from the system, however, being reinstalled with half stiffnes from 11 s.

2.17 ELLOAD - Element loads

The element loads are specified by the following format:

ELLOAD HIST DIR ELNR1 LOAD [ELNR2 LOAD2]
[REPEAT N ELINC]

where

HIST: Load history number.

DIR: Element load direction relative to the element axis **1-3** = load along x-z, **4-6** = moment about x-z. Note that for elements of type HELIX2XX, **ELLOAD** can only be applied with **DIR**=**3**.

ELNR1: Element ID of first element.

LOAD: Load for first element.

ELNR2: Element ID of last element.

LOAD2: Load for last element. Linear load interpolation is applied for the intermediate elements.

If the **REPEAT** command is introduced then the previous sequence of load generation is repeated:

N: Number of times to repeat.

ELINC: Element increment.

The **ELLOAD** command applies only for the PIPE, CABLE and SPRING element types, and will be a dummy command if applied for other element types.

The following distributed loads are applied for PIPE and CABLE elements:

Value of **DIR**:	**LOAD1/LOAD2** is:
1	force per length along local x axis (unit: FL^{-1})
2	force per length along local y axis (unit: FL^{-1})
3	force per length along local z axis (unit: FL^{-1})
4	moment per length about local x axis (unit: FLL^{-1})
5	moment per length about local y axis (unit: FLL^{-1})
6	moment per length about local z axis (unit: FLL^{-1})

The following concentrated loads are applied at the first element node for SPRING elements:

Value of **DIR**:	**LOAD1/LOAD2** is:
1	force in local x axis (unit: F)
2	force in local y axis (unit: F)
3	force in local z axis (unit: F)
4	moment about local x axis (unit: FL)
5	moment about local y axis (unit: FL)
6	moment about local z axis (unit: FL)

EXAMPLE:

```
#       hist  dof  elnr1  load1  elnr2  load2
ELLOAD  100   1    3      1.0    101    2.0
```

2.18 ELMASS - Element mass properties

Element mass matrices, m_0, can be specified. Element mass coefficients m_{ii} are given for each DOF in a node. The unit depends on element type, for PIPE and soil CONT elements the coefficients are per unit length (units: ML^{-1} for translation and ML for

rotation), for SPRING, CONT124, 130, 164, 170 lumped coefficients are given (units: M and ML^2).

The following format is applied.

ELMASS GRNAME TYPE M1 M2 M3 M4 M5 M6

where

GRNAME: Element group name.

TYPE: Element property type which may have only one option **BEAM** so far. The **BEAM** option covers the PIPE and SPRING element types.

M1: Mass coefficient for local x element direction.

M2: Mass coefficient for local y element direction.

M3: Mass coefficient for local z element direction.

M4: Mass coefficient for rotation acocciated to local x element direction.

M5: Mass coefficient for rotation acocciated to local y element direction.

M6: Mass coefficient for rotation acocciated to local z element direction.

For two-noded elements, the same mass coefficients are applied in both element ends. For the PIPE elements the resulting mass matrix is added to the mass matrix obtained from the **ELPROP** dry weight data. The **ELMASS** data are only applied in dynamic analyses.

EXAMPLE:

```
#      grname  type m1       m2       m3       m4     m5      m6
ELMASS vessel1 beam 0.0076e0 0.0076e0 0.0076e0 1.00e0 1.000e0 1.000e0
```

2.19 ELORIENT - Orientation of elements

The **ELORIENT** command is used to define the initial orientation of the element coordinate systems, and must be given. For the PIPE and HSHEAR elements, the orientations are defined by specifying the position of the xy-plane of the local element system relative to the global coordinate system. This is done by defining the position vector $\mathbf{R}$ in global coordinates of one point in the local xy-plane, see Fig. 2.23.

For the HELSPR437, CONT, BODY and SPRING elements, the orientation is defined by specifying a set of three consecutive Tait–Bryan angles (θ_x, θ_y, θ_z) in global coordinates that rotates the element coordinate system relative to the coordinate system of the node specified as **NOD1** in the **ELCON** card, see Section 2.13. The relations between

the global, nodal and element coordinates are shown in Fig. 2.24. As illustrated in Fig. 2.24, an initial nodal rotation specified by the **NOORIENT** card in Section 2.30 will also give a corresponding rotation of the element system relative to the global coordinate system.

Seabed contact elements shall be oriented with the initial x-axis coincident with the initial x-axis of the attached pipe element. Otherwise, the local x- and y-forces will not be directed along and transverse to the pipeline, respectively, and the local x-moment will not be acting about the pipeline x-axis, see Sections 2.26.9 and 2.26.14.

During the analysis, the element coordinate system for PIPE and HSHEAR elements follows the rotation of the end nodes in an averaged sense, while the element coordinate system for the HELSPR437, CONT, BODY and SPRING elements follows the rotation of the node specified as **NOD1** in the **ELCON** card.

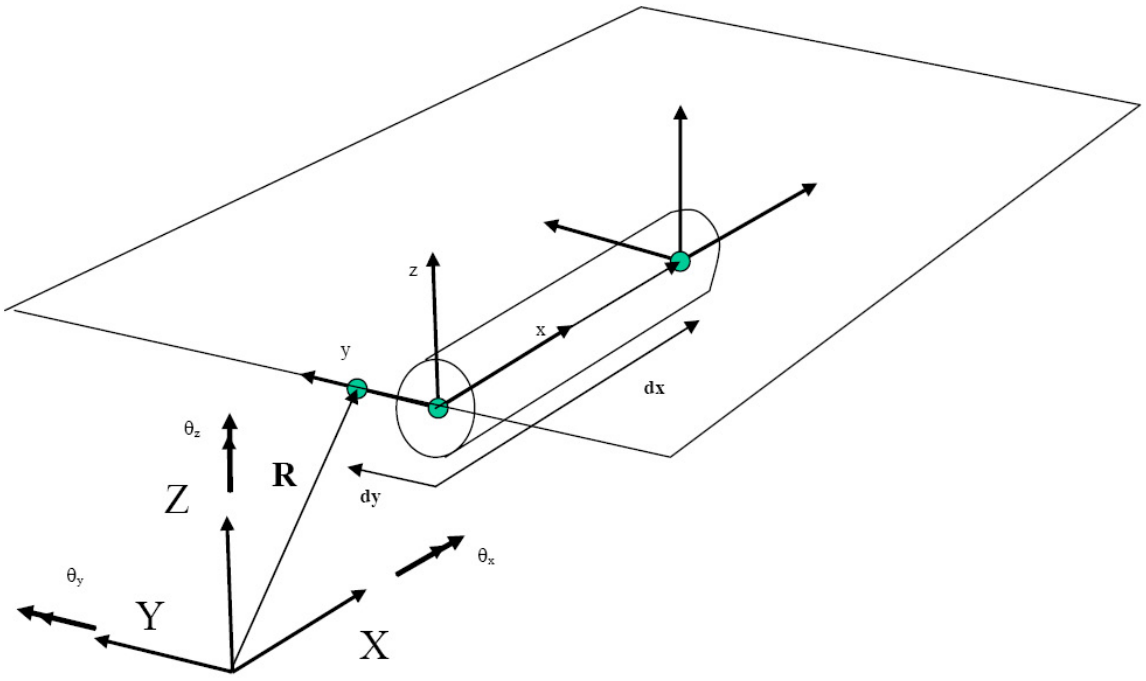


Figure 2.23: **ELORIENT** command for PIPE and HSHEAR elements.

The format is similar to the **NOCOOR** command, except that element ID numbers are used:

```
ELORIENT TYPE ELID X Y Z
                    [..    .. .. ..
[REPEAT N INC XINC YINC ZINC]
```

where

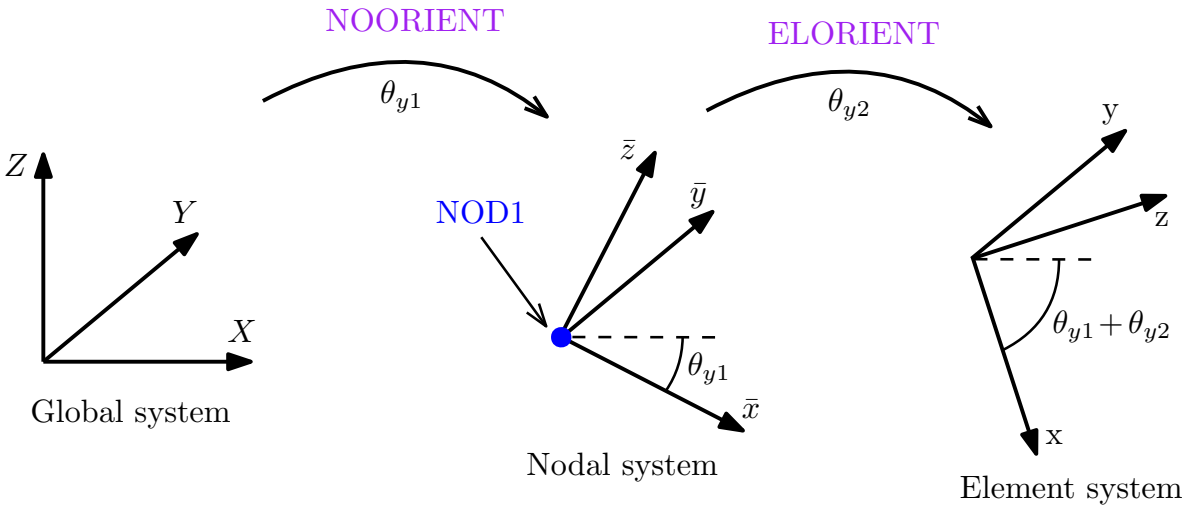


Figure 2.24: **ELORIENT** command for the HELSPR437, CONT, BODY and SPRING elements.

TYPE: Type of orientation. This parameter can have the values **COORDINATES** or **EULERANGLE**.

In case of **COORDINATES** the remaining parameters have the following meaning:

ELID: id of element.

X: Global x-coordinate of the position vector **R**.

Y: Global y-coordinate of the position vector **R**.

Z: Global z-coordinate of the position vector **R**.

By including the **REPEAT** command the **ELORIENT COORDINATES** sequence is repeated **N** times:

N: number of repetitions

INC: element increment

XINC: Global x-coordinate increment

YINC: Global y-coordinate increment

ZINC: Global z-coordinate increment

whereas if **TYPE** = **EULERANGLE**:

ELID: id of element.

X: Tait–Bryan angle around local x-axis.

Y: Tait–Bryan angle around local y-axis.

Z: Tait–Bryan angle around local z-axis.

By including the **REPEAT** command the **ELORIENT EULERANGLE** sequence is repeated **N** times:

N: number of repetitions

INC: element increment

XINC: Tait–Bryan angle increment around local x-axis

YINC: Tait–Bryan angle increment around local y-axis

ZINC: Tait–Bryan angle increment around local z-axis

In both cases an arbitrary sequence may be given, as long as **ELEMENT** is increasing. If the difference between two consequtive nodes is larger than 1, then linear interpolation is used to create intermediate element orientation.

EXAMPLE:

```
#
#         type             elemntnr      x     y       z
ELORIENT  COORDINATES         1         0.0   1.0     0.0
                             11         0.0   1.0   -10.0
                             21         0.0   1.0   -15.0
                             40         0.0   1.0   -19.75
#
#         n    inc  dx  dy  dz
REPEAT    3     40  0   0  -20
#
#
#        type           elementnr    tx    ty      tz
ELORIENT EULERANGLE       3001      0     0       0
```

2.20 ELPROP - Element properties

The following element property types are available:

PIPE : Properties for the PIPE element types

BELLMOUTH : Bellmouth contact properties

COMPIPE : Properties for COMPIPE42.

CABLE : Properties for the CABLE element types

ROLLER : Roller properties (element types CONT124 and CONT164)

BODY : Properties of body object

GENSPRING : User defined non-linear spring properties for SPRING137

SHEARHELIX : Shear element properties for HSHEAR353 and the tubular option for HSHEAR363 and HSHEAR364.

SHEAR2HELIX : Shear element properties for the helix option for HSHEAR363 and HSHEAR364.

SHEARMODEL : Shear interaction model selection. This enables the user to take into account the shear flexibility of the antiwear tape related to the element types PIPE52 and HSHEAR352 when using ITCODE and FLEXCROSS, see Section 2.10. Note that when applying 353FLEXCROSS, this relates to the element types HCONT453, HCONT454 and HCONT463.

LAYERCONTACT : Layer contact element properties for HCONT453, HCONT454, HCONT463 HCONT464 and HCONT473. The input is only necessary if the cross-section is manually defined, i.e. not using 353FLEXCROSS, see Section 2.10.

HELSPRING : User defined non-linear spring properties for HELSPR437.

HELIX : Helix element properties for HELIX231 and HELIX233.

SOILCONTACT : Properties for time-controlled hyper-elastic springs. Optional element property type only allowed for CONT126.

Each element property type is defined by the following format:

```
ELPROP ELGRP EPTYP ...
```

where

ELGRP: Element group name.

EPTYP: Element property type as defined above.

The number and definition of the following parameters depend on the choice of element property type.

2.20.1 PIPE

```
ELPROP ELGRP PIPE RAD TH CDR CDT CMR CMT MD MS ODP ODW RKS
[PHIST MHIST ELTIME=VALUE AUTOPLAST=VALUE SHEARA=VALUE SC-
FAX=VALUE SCFMY=VALUE SCFMZ=VALUE]
```

RAD: Structural radius i.e. the mean radius of the pipe wall (unit: L).

TH: Structural thickness, i.e. the wall thickness (unit: L).

CDR: Radial drag coefficient (unit: -).

CDT: Tangential drag coefficient (unit: -).

CMR: Radial mass coefficient. Must be ≥ 1.0. Normally equal to 2.0 for circular pipes which results in an added mass of $(\mathbf{RMADD} - 1.0)\rho_w \frac{\pi}{4} D^2$ (unit: -).

CMT: Tangential mass coefficient. Must be ≥ 1.0. A value of 1.0 results in zero tangential added mass according to $(\mathbf{RMADD} - 1.0)\rho_w \frac{\pi}{4} D^2$ (unit: -).

MD: Dry mass (unit: ML^{-1}).

MS: Submerged mass = dry mass - buoyancy mass (unit: ML^{-1})

ODP: Outer diameter D_{op} (unit: L)

ODW: External wrapping outer diameter D_{ow} (unit: L)

RKS: External wrapping fraction η (Range: 0-1). Hence the hydrodynamic diameter that will be applied to calculate drag and mass forces will be: $D = (1-\eta)D_{op} + \eta D_{ow}$. Note that the torsion rotational inertia will be calculated as $\rho\frac{\pi}{32}D_{op}^4$

PHIST: External pressure and buoyancy mass history ID number, optional.

MHIST: Dry mass history ID number, optional.

ELTIME=VALUE: This option is only relevant for PIPE33 and PIPE34. For PIPE33, the activation time for the elasto-plastic material model is specified by **VALUE**, i.e. the elements behave elastically until **VALUE** is exceeded. For PIPE34, the activation time for the initial bend curvature is specified by **VALUE**. The stress free bend radius need to be initiated by the **INISTR** card and completed at time=**VALUE**, see Section 2.25, optional (unit: T).

AUTOPLAST=VALUE: By specifying **AUTOPLAST=1**, the elements behave elastically until the proportionality limit is reached. The feature is not active if **VALUE≠1**. Only available for PIPE33, optional.

SHEARA=VALUE: Shear deformations with a shear rigidity (shear area times shear modulus) given by **VALUE** will be applied. An infinite shear stiffness results for **VALUE=0**. Only available for PIPE31, optional (unit: L^2).

SCFAX=VALUE: **VALUE** is the stress concentration factor used when calculating the axial stress. Only available for PIPE31, optional (unit: -).

SCFMY=VALUE: **VALUE** is the stress concentration factor used when calculating the bending stress from moment about local y-axis. Only available for PIPE31, optional (unit: -).

SCFMZ=VALUE: **VALUE** is the stress concentration factor used when calculating the bending stress from moment about local z-axis. Only available for PIPE31, optional (unit: -).

The **ELTIME=VALUE** and **AUTOPLAST=VALUE** options work such that no plasticity is considered until the time specified by **ELTIME=VALUE** is exceeded. If both quantities are nonzero this is still the case, however, only elements that actually are plastified after the given time will behave plastic. This option may speed up pipeline buckling analysis with a factor 5-10.

For the **ELASTOPLASTIC** material type, **RAD** and **TH** are employed for calculation of the cross-section stiffness, whereas for other material types the stiffness is taken directly from the **MATERIAL** input card in Section 2.26.

Regarding contact search, the inner diameter of the pipe is set equal to 2***RAD** - **TH**. The outer contact diameter OD_c is taken as the maximum of the outer pipe diameter and the wrap diameter according to $OD_c = \max(\mathbf{ODP}, \mathbf{ODW})$.

EXAMPLE:

```
#       name       eptyp  rad     th      CDr  Cdt  CMr  CMt  wd   ws   ODp   ODw  rks
ELPROP  ormpipe1   pipe   0.3221  0.0346  0.8  0.1  2.0  1.1  5.6  1.7  0.67  0.67 0.5
#
# phist  mhist  eltime=value  autoplast=value  sheara=value  scfax=value  scfmy=value
100      200    eltime=10.0   autoplast=1      sheara=0.0    scfax=1.1    scfmy=1.5
#
# scfmz=value
scfmz=1.5
```

2.20.2 COMPIPE

ELPROP ELGRP COMPIPE RADE RADI CDR CDT CMR CMT MD MS ODP ODW RKS [PHIST MHIST ELTIME=VALUE AUTOPLAST=VALUE TCURV=VALUE]

RADE: External radius where external pressure acts (unit: L).

RADI: Internal radius where internal pressure acts (unit: L).

CDR: Radial drag coefficient (unit: -).

CDT: Tangential drag coefficient (unit: -).

CMR: Radial mass coefficient. Must be ≥ 1.0. Normally equal to 2.0 for circular pipes which results in an added mass of $(\mathbf{RMADD} - 1.0)\rho_w \frac{\pi}{4} D^2$ (unit: -).

CMT: Tangential mass coefficient. Must be ≥ 1.0. A value of 1.0 results in zero tangential added mass according to $(\mathbf{RMADD} - 1.0)\rho_w \frac{\pi}{4} D^2$ (unit: -).

MD: Dry mass (unit: ML^{-1}).

MS: submerged mass = dry mass - buoyancy mass (unit: ML^{-1})

ODP: Outer diameter D_{op} (unit: L)

ODW: External wrapping outer diameter D_{ow} (unit: L)

RKS: External wrapping fraction η (Range: 0-1). Hence the hydrodynamic diameter that will be applied to calculate drag and mass forces will be: $D = (1 - \eta)D_{op} + \eta D_{ow}$

PHIST: External pressure and buoyancy mass history ID number, optional.

MHIST: Dry mass history ID number, optional.

ELTIME=VALUE: The activation time for the elasto-plastic material model is specified by **VALUE**, i.e. the elements behave elastically until **VALUE** is exceeded. Only available for COMPIPE42, optional (unit: T).

AUTOPLAST=VALUE: By specifying **AUTOPLAST=1**, the elements behave elastically until the proportionality limit is reached. The feature is not active if **VALUE≠1**. Only available for COMPIPE42, optional.

TCURV=VALUE: The activation time for the bending model is specified by **VALUE**. Gives zero bending internal moments and stiffness for zero moment until **VALUE** is exceeded. Only available for COMPIPE42, optional (unit: T).

The **ELTIME=VALUE** and **AUTOPLAST=VALUE** options work such that the elements behaves elastic until the time specified by **ELTIME=VALUE** is exceeded. If both quantities are nonzero this is still the case, however, only elements that actually are plastified after the given time will behave plastic.

Regarding contact search, the inner diameter of the pipe is set equal to 2*__RAD__ - __TH__. The outer contact diameter OD_c is taken as the maximum of the outer pipe diameter and the wrap diameter according to $OD_c = \max(\mathbf{ODP}, \mathbf{ODW})$.

EXAMPLE:

```
#      elgrp       eptyp     rade  radi  Cdr  Cdt  CMr  CMt  md   ms   ODp  OD   rks
ELPROP restrictor compipe   0.11  0.10  0.8  0.0  2.0  1.0  0.4  0.1  0.1  0.1  1.0
#
# phist  mhist  eltime=value  autoplast=value  tcurv=value
100      200    eltime=2.0    autoplast=1      tcurv=4.0
```

2.20.3 BELLMOUTH

ELPROP ELGRP BELLMOUTH DP [INSIDE GEOFAC]

DP: Pipe diameter, inside or outside depending on the meaning of the inside parameter defined below (unit: L).

INSIDE: Parameter where **1** means that the master is inside the slave group of larger diameter, and any other value means that the master is outside the slave group. Optional, default value: 1

GEOFAC: Scaling factor for the geometric stiffness. A value greater than **1.0** can be applied to stabilize the pipe-bellmouth interaction behaviour. Optional, default value: 1.0

If the master is inside the slave pipe, **DP** is the outside pipe diameter of the master pipe. Otherwise **DP** is the inside diameter of the master pipe.

The **GEOFAC** scaling factor is only applied for the geometric stiffness associated with direction change of the contact point normal vector. The contact force at equilibrium

will not be affected by **GEOFAC**.

EXAMPLE:

```
#       elgrp name   type       dp   inside  geofac
ELPROP  bellcontact  bellmouth  0.4  1       1.0
```

2.20.4 BODY

ELPROP ELGRP BODY GEOM WS WD WDTHX WDTHY WDTHZ CDX CDY CDZ CDTHX CDTHY CDTHZ CMX CMY CMZ CMTHX CMTHY CMTHZ COGX COGY COGZ [PHIST MHIST ICCHYD HCX HCY HCZ]

GEOM: Body geometry name, defined by the card **GEOM**.

WS: Submerged mass = dry mass - buoyancy mass (unit: M).

WD: Dry mass (unit: M).

WDTHX: X-rotation structural mass at COG (unit: ML^2).

WDTHY: Y-rotation structural mass at COG (unit: ML^2).

WDTHZ: Z-rotation structural mass at COG (unit: ML^2).

CDX: Drag coefficient for x-direction (unit: L^2).

CDY: Drag coefficient for y-direction (unit: L^2).

CDZ: Drag coefficient for z-direction (unit: L^2).

CDTHX: Drag coefficient for x-rotation at hydrodynamic center (unit: L^5).

CDTHY: Drag coefficient for y-rotation at hydrodynamic center (unit: L^5).

CDTHZ: Drag coefficient for z-rotation at hydrodynamic center (unit: L^5).

CMX: Added mass coefficient for x-direction (unit: L^3).

CMY: Added mass coefficient for y-direction (unit: L^3).

CMZ: Added mass coefficient for z-direction (unit: L^3).

CMTHX: Added mass coefficient for x-rotation at hydrodynamic center (unit: L^5).

CMTHY: Added mass coefficient for y-rotation at hydrodynamic center (unit: L^5).

CMTHZ: Added mass coefficient for z-rotation at hydrodynamic center (unit: L^5).

COGX: Local x-eccentricity from element origin to COG (unit: L).

COGY: Local x-eccentricity from element origin to COG (unit: L).

COGZ: Local x-eccentricity from element origin to COG (unit: L).

PHIST: External pressure and buoyancy mass history ID number, optional.

MHIST: Dry mass history ID number, optional.

ICCHYD: Hydrodynamic Coriolis-centripetal load switch, optional, default value: 0.

0 : No hydrodynamic Coriolis-centripetal loads are included.

1 : Hydrodynamic Coriolis-centripetal loads are included.

HCX: Local x-eccentricity from element origin to hydrodynamic center, optional, default value: **COGX**(unit: L).

HCY: Local y-eccentricity from element origin to hydrodynamic center, optional, default value: **COGY**(unit: L).

HCZ: Local z-eccentricity from element origin to hydrodynamic center, optional, default value: **COGZ**(unit: L).

The center of buoyancy coincides with the COG location.

With reference to **ICCHYD**=**1**, the hydrodynamic Coriolis-centripetal loads consist of added mass forces and moments in DOFs 1-6 due to body angular velocity and Munk moment loads in DOFs 4-6 due to linear velocity. The former load type may be important for large angular velocities combined with large hydrodynamic mass variation between the DOFs, such as for a trawl board colliding with a pipeline.

EXAMPLE:

```
#       name type  geom  ws   wd   wdthx wdthy wdthz cdx  cdy  cdz  cdthx cdthy cdthz
ELPROP sdd  body  geo1  2.0  2.1  0.0   0.0   0.0   1.5  1.5  1.5  0.0   0.0   0.0
# cmx  cmy  cmz  cmthx cmthy cmthz cogx cogy cogz phist mhist icchyd hcx hcy hcz
 0.15  0.15 0.15 0.0   0.0   0.0   0.0  0.0  0.0    100   110    1    0.1 0.0 1.0
```

2.20.5 CABLE

ELPROP ELGRP CABLE CDR CDT CMR CMT MD MS ODP ODW RKS [PHIST MHIST]

CDR: Radial drag coefficient (unit: -).

CDT: Tangential drag coefficient (unit: -).

CMR: Radial mass coefficient (Normally 2.0 for circular pipes. This will result in an added mass of $(\text{RMADD} - 1.0)\rho_w \frac{\pi}{4} D^2$) (unit: -).

CMT: Tangential mass coefficient (Must be $\geq$ 1.0. A value of 1.0 will result in no tangential added mass according to the above formula.) (unit: -).

MD: Dry mass (unit: ML^{-1}).

MS: Dry mass - buoyancy mass (unit: ML^{-1}).

ODP: Outer diameter D_{op} (unit: L)

ODW: External wrapping outer diameter D_{ow} (unit: L)

RKS: External wrapping fraction η (Range: 0-1). Hence the hydrodynamic diameter that will be applied to calculate drag and mass forces will be: $D = (1-\eta)D_{op} + \eta D_{ow}$

PHIST: External pressure and buoyancy mass history ID number, optional.

MHIST: Dry mass history ID number.

The outer contact diameter OD_c is taken as the maximum of the outer diameter and the wrap diameter according to $OD_c = \max(\textbf{ODP}, \textbf{ODW})$.

EXAMPLE:

```
#
#      elgrpname elproptype Cdr Cdt CMr CMt wdry   wsub   ODpipe ODwrap extwrapfrac
ELPROP wire      cable      0.8 0   2.0   1.0   0.0395 0.01   0.1    0.1    0
```

2.20.6 ROLLER

ELPROP ELGRP ROLLER RD [CONTPAR1=VALUE CONTPAR2=VALUE]

RD: Roller diameter (unit: L).

CONTPAR1=VALUE: **CONTPAR1=** is a character string and **VALUE** is an integer equal to either **0** or **1**. For **VALUE=0** which is default, an arbitrary number of contact elements may obtain contact with the same pipe element. If **VALUE=1**, only one contact element can be active for each pipe element.

CONTPAR2=VALUE: **CONTPAR2=** is a character string and **VALUE** is an integer that defines the number of pipe elements a contact point is allowed to move along during one increment or iteration step before the contact is deactivated. For the example in Fig. 2.25, **VALUE=2** means that contact search will be performed for pipe element i which had active contact at the previous load step and for its 4 neighbouring elements $i-2$, $i-1$, $i+1$ and $i+2$. If contact is not present at any of the 5 pipe elements at the converged equilibrium state, e.g. if the contact point has moved to elements $i-3$ or $i+3$, the contact status will be set to inactive and the friction variables will be reset. By default **VALUE=1** is applied. The **CONTPAR2=VALUE** option is currently only available for CONT164 in combination with the **ISOKXYCONTACT** material type in Section 2.26.11.

The **CONTPAR1=VALUE** option is applied in situations where two or more contact elements may detect the exact same pipe contact point. A typical example is shown in Fig. 2.26, where nine CONT164 elements with identical rollers are applied to handle the pipe contact. If **CONTPAR1=0** is applied, all of the contact elements will detect

the same contact point, i.e. the one with largest interpenetration, resulting in too concentrated contact forces and too large penetrations at the locations where contact is falsely not detected. By instead using **CONTPAR1=1**, the contact element that first detects a contact point, will tag the pipe element with a contact indicator so that the remaining contact elements cannot be activated for the given pipe element.

Note that when **CONTPAR1=1**, the contact element group still detects contact with a pipe element that interacts simultaneously with a contact element group which has been assigned **CONTPAR1=0**. Likewise, a contact element group with **CONTPAR1=0** will detect contact at a pipe element that simultaneously interacts with a contact element group which has been assigned **CONTPAR1=1**. Hence, if a pipe element shall be fully restricted to only have one roller contact point, **CONTPAR1=1** must be assigned to all contact element groups that possibly can come into contact with the pipe element. Note also that the **CONTPAR1=VALUE** functionality only affects roller-based contact elements. Hence, a CONT164 element with **CONTPAR1=1** will detect contact at a pipe element that simultaneously interacts with a CONT152 element.

The **CONTPAR2=VALUE** option is used for avoiding that unrealistic tangential pipe contact point displacements occur during a load step. For the analysis in Fig. 2.26, a single contact element may detect contact points that differs with an angular coordinate of up to 90 degrees between the load steps, resulting in completely wrong tangential displacement increments for updating the friction force. By selecting e.g. **CONTPAR2=1**, the issue is avoided as the contact will then be deactivated if the pipe contact point detected by the contact element moves more than 1 pipe element during the load step. However, in that case it must be assured that there are enough contact elements available to detect the occurrence of new contact points, otherwise contact will falsely not be detected.

The input data for the example in Fig. 2.26 is as follows:

EXAMPLE:

```
#       name    type     rd
ELPROP  roll1   roller   0.5   contpar1=1   contpar2=1
#
#
#       name    eptyp   matname   elid    nod1             n  nelinc nodinc
ELCON   roll1   cont164 rollmat   40001   5301     repeat  9     1     0
#     type    elid  elend  xecc  yecc  zecc  yphi  dx1  dy1  dz1  dx2  dy2  dz2
ELECC stinger 40001   1     0.0   0.0   0.0   0.0  0.0  1.0  0.0  0.0  2.0  0.0
#      n  elinc  ds  dyphi
repeat 9    1     0    0.0
```

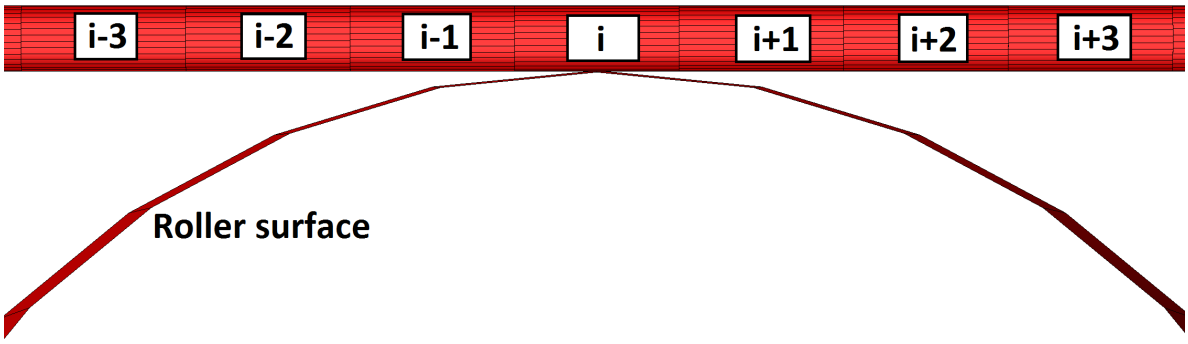


Figure 2.25: Defining maximum allowable incremental pipe contact point displacement by **CONTPAR2=VALUE**

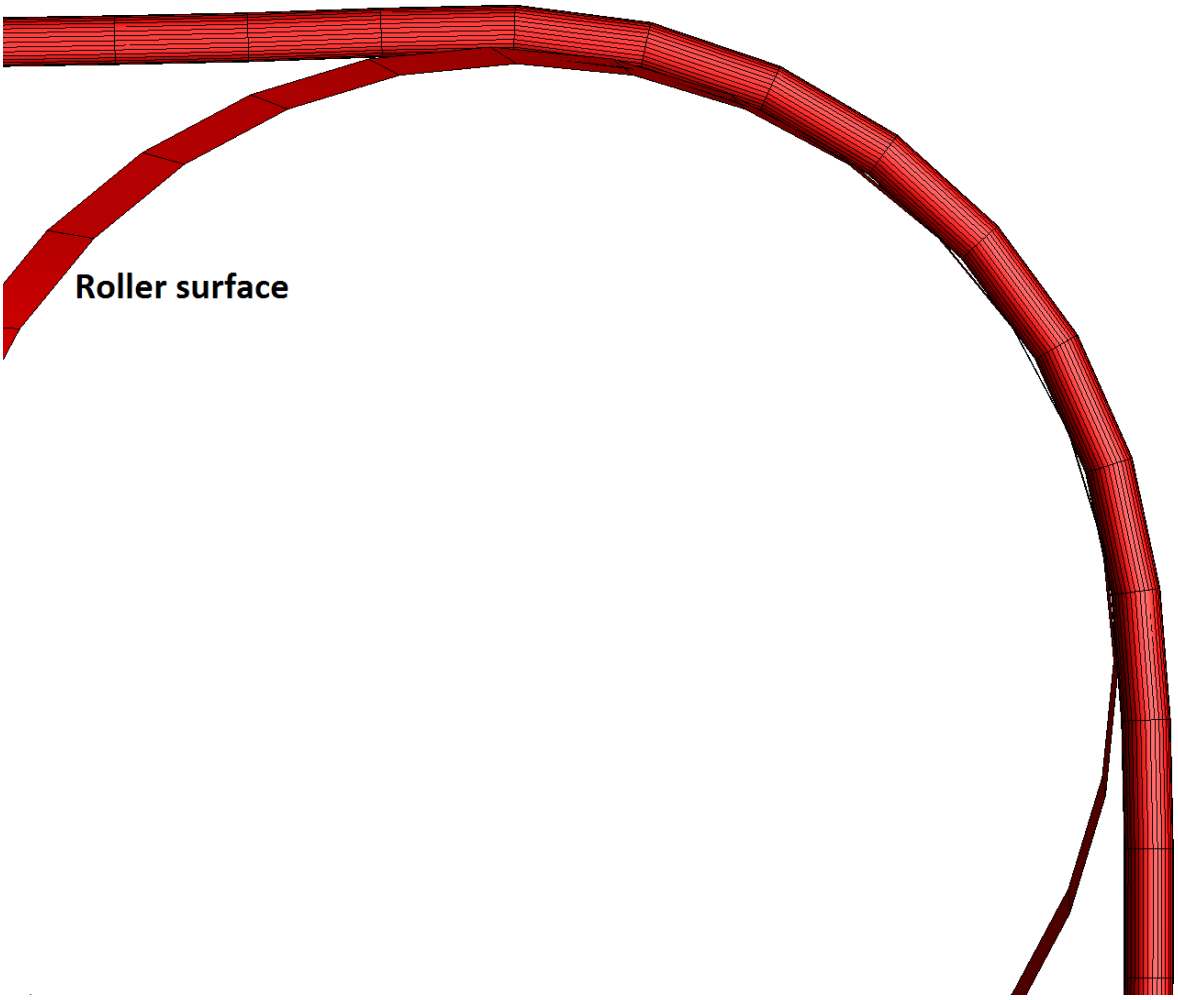


Figure 2.26: Analysis where **CONTPAR1=1** and **CONTPAR2=0** or **1** is recommended

2.20.7 GENSPRING

ELPROP ELGRP GENSPRING TX TY TZ RX RY RZ [IREF ICOULCNTR]

TX: Time step at which the x-spring is activated.

TY: Time step at which the y-spring is activated.

TZ: Time step at which the z-spring is activated.

RX: Time step at which the x-rotation spring is activated.

RY: Time step at which the y-rotation spring is activated.

RZ: Time step at which the z-rotation spring is activated.

IREF: Integer parameter for controlling the reference coordinate system applied for the material curves, the element displacements and the element forces. Further description of allowable values are given below.

ICOULCNTR: Integer parameter for controlling the **COULOMB** force scaling option specified by the **GENSPRING** material type in Section 2.26.16. Further description of allowable values are given below.

For **IREF**$\neq$ **1**, the material curves, the element displacements and the element forces refer to the element coordinate system. This is the default behavior when the optional parameter **IREF** is omitted. During the analysis, the element coordinate system will follow the rotations of element node number 1, see **NOD1** for the SPRING element defined by the **ELCON** card in Section 2.13.

For **IREF**=**1**, the material curves, the element displacements and the element forces refer to the global coordinate system. The only exception from this is when a special reference system is used for the nodal motion of element node 1, see **NOD1** for the SPRING element defined by the **ELCON** card in Section 2.13. A special nodal motion reference system will be applied for **NOD1** in the following cases:

- When the node is assigned the boundary condition types **LOCAL** or **SPECIAL**, see the **BONCON** card in Section 2.2. In that case, the material curves, the element displacements and the element forces refer to the local or special coordinate system of the node, which rotates together with the node during its motion.

- When the node is a helix node. This is the case if the node also is defined as element node number 3 or 4 for the HSHEAR352 and HSHEAR353 element types, see **NOD3** and **NOD4** defined by the **ELCON** card in Section 2.13. Then the material curves, the element displacements and the element forces refer to the local coordinate system of the helix node.

For **ICOULCNTR**$\neq$ **1**, the x-direction force F_x is calculated by performing **COULOMB** scaling of the force f_x from the x-direction material curve according to $F_x = f_x\sqrt{F_y^2 + F_z^2}$,

where F_y and F_z are the spring forces in the y- and z-directions, respectively. The moment about the x-axis is calculated as $M_x = m_x\sqrt{F_y^2 + F_z^2}$, where m_x is taken from the x-rotation material curve. Note that these scaling conventions means that the friction coefficient μ must be built into the dimensionless force f_x specified for the x-direction material curve, and for the x-rotation material curve both the friction coefficient μ and the distance to the modelled contact point r must be incorporated into the moment quantity m_x which has unit L. The **COULOMB** force scaling option is not applied in the y-direction for **ICOULCNTR**$\neq$ **1**, i.e. the y-direction force F_y is set equal to f_y specified for the y-direction material curve. By default the option **ICOULCNTR**$\neq$ **1** is applied when the parameter **ICOULCNTR** is omitted.

By selecting **ICOULCNTR**= **1**, a 2D isotropic friction model is applied by performing **COULOMB** scaling of f_x from the x-direction material curve. Here, the force f_y specified for the y-direction material curve is dummy, and the 2D friction force in the xy-plane is given by $\mathbf{F}_t = f_x|F_z|\mathbf{t}$ where $\mathbf{t}$ is a 2D unit tangent vector in the sliding direction. Note that this scaling convention means that the friction coefficient μ must be built into the dimensionless force f_x specified for the x-direction material curve. The **COULOMB** scaling option is not applied for rotation about the x-axis for **ICOULCNTR**= **1**, i.e. the moment M_x about the x-axis is set equal to the moment m_x specified for the x-rotation material curve.

The parameter **ICOULCNTR** is dummy when the default **USERDEFINED** force scaling option in Section 2.26.16 is applied.

Note that the **COULOMB** scaled material curves should be defined by the **EPCURVE** material curve option in Section 2.26.4, otherwise reversal of the friction force will not be correctly modelled. A typical application for **ICOULCNTR**$\neq$ **1** is modelling of axial friction for pipe-in-pipe problems where the force $\sqrt{F_y^2 + F_z^2}$ represents the radial force.

EXAMPLE:

```
#      name      type         tstpx  tstpy  tstpz  tstprx  tstpry  tstprz
ELPROP vessel1   genspring    1.0    1.0    1.0    1.0     1.0     1.0
#
# iref  icoulcntr
    0      1
```

2.20.8 HELSPRING

ELPROP ELGRP HELSPRING STEP

STEP: Load step at which the friction spring in local curvilinear helix system is activated.

2.20.9 HELIX

ELPROP ELGRP HELIX CONMAT RAD TH HELIXRAD LAYANG ANGSTART WD WS IFRIC [PHIST MHIST TTIME]

CONMAT: The name of the contact material describing the helix radial stiffness and the longitudinal helix friction properties

RAD: Structural radius i.e. the mean radius of the helix tube wall (unit: L).

TH: Structural thickness, i.e. the wall thickness of the helix tube (unit: L).

HELIXRAD: helix radius (unit: L).

LAYANG: Helix lay angle α, positive according to the right hand rule as in Fig. 2.27

ANGSTART: Angular position angle θ, see Fig. 2.27(unit: R).

WD: Dry weight (unit: ML^{-1}).

WS: Submerged weight (unit: ML^{-1}).

IFRIC: friction stress control parameter:

- **0 :** No friction stress analysis is carried out.
- **2 :** The contact pressure related to bending friction stresses are included in the total contact pressure. This is important for manufacturing stress evaluations where the back tension is small.
- **ELSE :** Friction stress analysis is carried out.

PHIST: External pressure and buoyancy mass history ID number, optional. Note that if **PHIST** is supplied, **MHIST** is also required.

MHIST: Dry mass history ID number.

TTIME: Time for tuning slip value for friction material in x-direction. Note that only the slip level of the curve is changed, all other points in the curve remain as defined by the user.

2.20.10 SZHELIX

ELPROP ELGRP SZHELIX CONMAT RAD TH HELIXRAD LPITCH NUSUAL NBACK DELTA ANGSTART LSTART WD WS IFRIC [PHIST MHIST TTIME]

CONMAT: The name of the contact material describing the helix radial stiffness and the longitudinal helix friction properties

RAD: Structural radius i.e. the mean radius of the helix tube wall (unit: L).

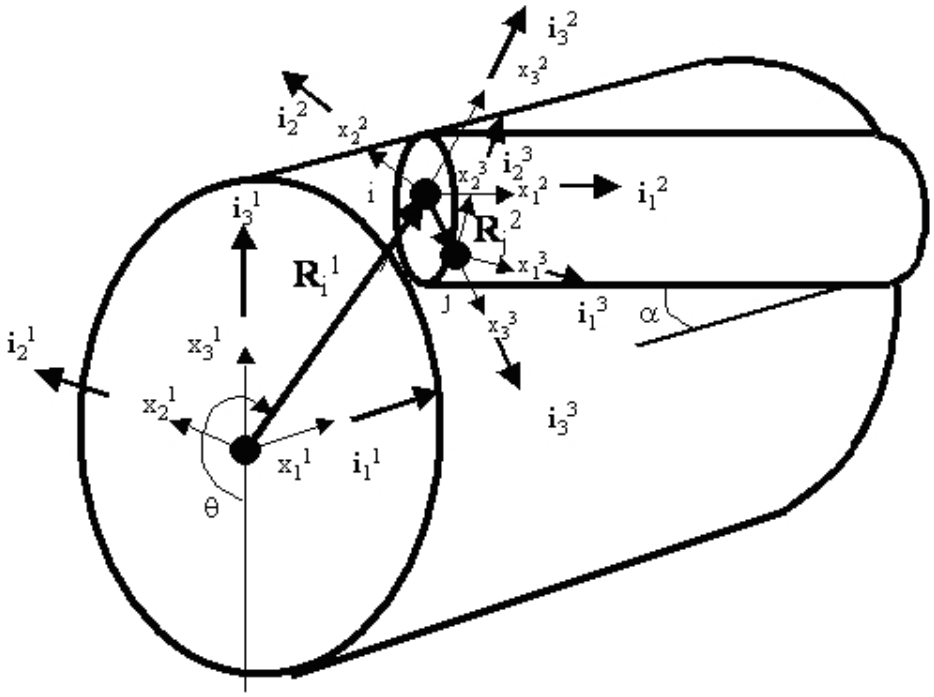


Figure 2.27: Helix coordinate system.

TH: Structural thickness, i.e. the wall thickness of the helix tube (unit: L).

HELIXRAD: helix radius (unit: L).

LPITCH: Pitch length according to the rule as defined in Fig. 2.28, i.e. the pitch length of one helix section (unit: L). The Sz curve is divided into periodic sections each including two helix curves and two turning curves. The length of each helix curve is **NUSUAL** × **LPITCH** and the length of each turning curve is **NBACK** × **LPITCH**.

NUSUAL: Number of ordinary helix pitches between turning direction, see Fig. 2.28.

NBACK: Number of ordinary helix lengths used for turning direction, see Fig. 2.28.

DELTA: The δ parameter according to Fig. 2.28, i.e. the length over which the gradient of lay angle change is to be taken, providing that the derivative of lay angle is continuous.

ANGSTART: Angular position angle θ, see Fig. 2.28(unit: R).

LSTART: The parameter lstart (unit: -), see Fig. 2.28. This parameter defines where in the curve the first nodal point starts. If it is zero BFLEX2010 assumes that the curve starts at the mid position of the section that has positive lay angle. Value 0.5 corresponds to mid position of the section of negative lay angle.

WD: Dry weight (unit: ML^{-1}).

WS: Submerged weight (unit: ML^{-1}).

IFRIC: friction stress control parameter:

0 : No friction stress analysis is carried out.

ELSE : Friction stress analysis is carried out.

PHIST: External pressure and buoyancy mass history ID number, optional. Note that if **PHIST** is supplied, **MHIST** is also required.

MHIST: Dry mass history ID number.

TTIME: Time for tuning slip value for friction material in x-direction. Note that only the slip level of the curve is changed, all other points in the curve remain as defined by the user.

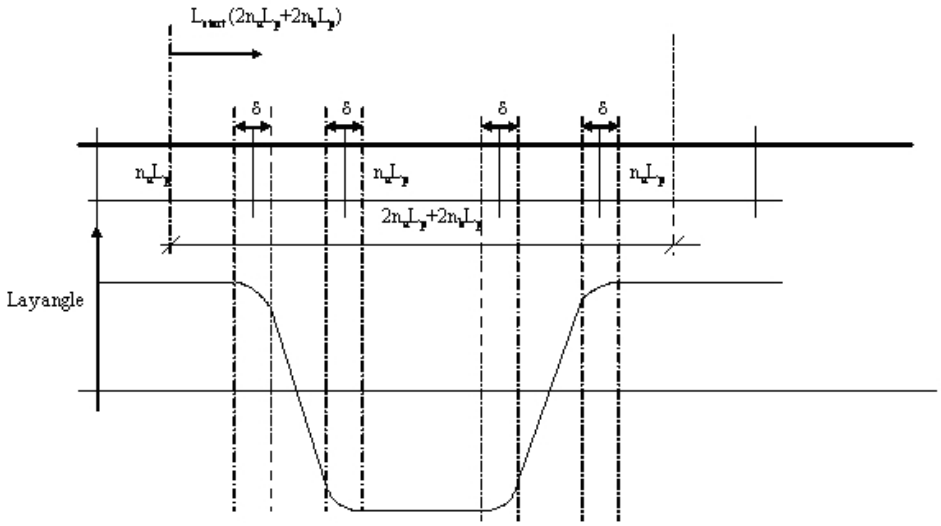


Figure 2.28: Szhelix coordinate system.

This input is only required during manual modelling without applying the FLEXCROSS features, see Section 2.10.1 and Section 2.10.2

2.20.11 LAYERCONTACT

ELPROP ELGRP LAYERCONTACT *GAP0 TTIME CNTR1 CNTR2 SCALEFACT* [KSHEAR]

GAP0: Initial gap. Default value =0.0, Unit: L. A non-zero value is only applicable for HCONT454. This must be given as the initial gap in the lateral transverse direction of the helix. For a tensile armour this is calculated as $g_0 = \frac{2\pi R cos\alpha}{n}(1 - F_f)$, where b is the wire width, F_f is the fill factor, R is the helix radius, α is the lay angle and n is the number of wires in the layer. It is noted that the gap is scaled by the SCALEFACT parameter such that no correction of the value is necessary if a tensile armour layer is represented by a reduced number of wires.

TTIME: The time when friction is activated. Default: 0.0 Unit: s

CNTR1: For HCONT454 this is the fill factor. i.e. the fraction filled by the cross-section of wires with respect to the layer circumference. Dummy for other HCONT elements

CNTR2: A contact search parameter that is only relevant for HCONT453 and HCONT464. Default:e 0.0 If set to 1 for HCONT453 an update contact search will be carried out one time prior to analysis. This is recommended if the contact element is used to describe contact between two armour layers. If set to 1 for HCONT464 the contact master (node 1 in the topology) is on the outside of the slave layer

SCALEFACT: Ratio between total number of helices in a layer and the number of helices used to represent that layer. (It is not necessary to represent all 70 tensile armour tendons rather use 10 and combine that with **SCALEFACT**=7). Default value 1.0

EXAMPLE:

```
#
#      elgrpname elproptype     gap0 ttime  cntr1 cntr2  scalefact
ELPROP layer     layercontact    D    D      D     1      5.0
```

2.20.12 SHEARHELIX

ELPROP ELGRP SHEARHELIX GEOTYPE W TH WD WS SCALEFACT [PHIST GHIST AXISYM]

GEOTYPE: Geometry type, **TUBE** or **RECTANGLE** for element type HSHEAR353, only **TUBE** for element type HSHEAR363. (unit: -).

In case of **TUBE** the parametes have the following meaning:

W: Structural (mean) radius (unit: L).
TH: Thickness (unit: L).

whereas if **TYPE** = **RECTANGLE**:

W: Width of rectangle (unit: L).
TH: Thickness (unit: L).

WD: Dry mass (unit: ML^{-1}).

WS: Submerged mass = Dry mass - Bouyancy mass (unit: ML^{-1}).

SCALEFACT: Ratio between total number of helices in a layer and the number of helices used to represent that layer (i.e. It is not necessary to represent all 70 tensile armour tendons rather use 10 and combine that with **SCALEFACT**=7)

PHIST: External pressure and buoyancy mass history ID.

GHIST: Gravitational history ID.

AXISYM: Allowing the user to control the shear interaction behaviour of the HSHEAR353 element. If **AXISYM** = 2 then the bending shear interaction is turned off, i.e. only axisymmetric shear interaction is included. For **AXISYM** =10, The axial stress from axisymmetric effects and bending are uncoupled. The axial stresses are determined without considing shear interaction, i.e. the stresses due to slip between layers only occur as a result of bending. The axisymmetric torsion and bending moment quantities are included. For **AXISYM** = 11, The same behaviour as described for **AXISYM**=10 is obtained, however, the axisymmetric torsion and bending moment quantities are neglected. Any other value of **AXISYM** will cause full interaction between all axisymmetric and bending effects. Default value = 10, i.e. fully coupled shear interaction is obtained

It is noted that this input is not necessary when using 353FLEXCROSS, see Section 2.10. However, it is still possible to overrule the **AXISYM** parameter = 10 (corresponding to uncoupling axisymmetric stress from bending stress) set by 353FLECROSS by applying **TUBE=NONE** and set all other parameters before **AXISYM** to D.

EXAMPLE:

```
#
#      elgrpname  elproptype geotype w      th     wd        ws        scalefact
ELPROP helixlayer shearhelix TUBE    0.3221 0.0346 0.5606e-3 0.1792e-3 0
```

2.20.13 SHEAR2HELIX

ELPROP ELGRP SHEAR2HELIX GEOTYPE W TH WD WS RAD ALFA SCALEFACT [PHIST GHIST]

GEOTYPE: Geometry type, **TUBE**, **RECTANGLE** or **TAPE** for element type HSHEAR363. (unit: -).

In case of **TUBE** the parametes have the following meaning:

R: Structural (mean) radius (unit: L).

TH: Thickness (unit: L).

whereas if **TYPE** = **RECTANGLE** or**TAPE**:

W: Width of rectangle (unit: L).

TH: Thickness (unit: L).

WD: Dry mass (unit: ML^{-1}).

WS: Submerged mass = Dry mass - Bouyancy mass (unit: ML^{-1}).

RAD: Helix radius (unit: L).

ALFA: Helix lay angle (unit: Radians).

SCALEFACT: Ratio between total number of helices in a layer and the number of helices used to represent that layer (i.e. It is not necessary to represent all 70 tensile armour tendons rather use 10 and combine that with **SCALEFACT**=7.0)

PHIST: External pressure and buoyancy mass history ID.

GHIST: Gravity history ID.

EXAMPLE:

```
#
#      elgrpname  elproptype  geotype   w     th    wd        ws        rad  alfa scalefact
ELPROP helixlayer shear2helix rectangle 0.010 0.003 0.5606e-3 0.1792e-3 0.05 0.60 7.0
```

2.20.14 SHEARMODEL

This element property relates to two modelling procedures enabled to describe a flexible pipe:

- By using FLEXCROSS and ITCODE, see Section 2.10
- By using 353FLEXCROSS , see Section 2.10

and concerns alternative friction models and possible softening in the pre-slip (stick) bending stiffness resulting from the fact that a flexible pipe include multible layers with different mechanical properties.

Previously, and by default, the stick stiffness was tuned based on the contact pressure obtained at the mean tension and pressure, activated by the user via the TIMEINI parameter, see Section 2.10.1 and Section 2.10.2. This results in a constant stick curvature/stick displacement which again means that the stick stiffness will depend on contact pressure variations due to dynamic tension, possibly causing numerical stability issues. This was not of much concern in the original version of BFLEX, which focused on fatigue analysis of North Sea risers, where the tension was typically a small fraction of the end cap force of a flexible riser during operation. However, to meet the requirements from deep water applications, a new friction model has been developed that is based on a constant stick stiffness as described here.

Then, it has been noted that the plane surfaces remain plane approach will in most cases overestimate the pre-slip (stick) bending stiffness. It is noted that the mostly

applied models ITCODE=21 & 31 and ITCODE=0 &1, respectively linked to the PIPE52 and HSHEAR352 tensile armour element groups (FLEXCROSS), already include some effect of shear interaction as they were calibrated versus experimental data. The same applies to the default friction models applied in 353FLEXCROSS via the contact elements HCONT453, HCONT454 & HCONT463. Therefore, the correct way of making use of the shear interaction stiffness parameter described here is by activating the new friction model and by starting out from the plane surface remain plane concept.

This means that for both the FLEXCROSS and 353FLEXCROSS options, by introducing the SHEARMODEL input for a given interface, the default friction model will (by default) be replaced by the new friction model to activate shear interaction, as described in the following.

Firstly and with reference to the original friction model, the slip curvature established at time TIMEINI, β_{2c0} is calculated as:

$$\beta_{2c0} = \frac{\mu(q_3^I + q_3^{I+1})}{EAcos^2\alpha sin\alpha} \tag{2.2}$$

where μ is the friction coefficient, q_3^I is the normal line load at interface I from contact pressure, EA is the axial stiffness of the wire, α is the helix lay angle.

which for the sandwich beam concept applied for HSHEAR352, is converted to a slip displacement as:

$$u_{1c} = \beta_{2c} R^2 \frac{cos^2\alpha}{sin\alpha} \tag{2.3}$$

where R is the helix radius.

Then by applying the moment-curvature option ITCODE=32 in FLEXCROSS and the old friction model, the slip curvature, β_{2c} (PIPE52) is modified by the shear stiffness parameter k as:

$$\beta_{2c} = [1 + \frac{sin^2\alpha EA}{kR^2}]\beta_{2c0} \tag{2.4}$$

For the new friction model, a constant shear interaction stick stiffness is applied. For the moment-curvature models obtained when the tensile armour is modelled by PIPE52 applying ITCODE21, ITCODE31 or ITCODE32, the plane surfaces remain plane stick bending stiffness, EI_{plane} is firstly calculated as:

$$EI_{plane} = \frac{4}{\pi^2} EAcos^3\alpha R^2 n \tag{2.5}$$

where n is the number of wires in the layer. Then the following modification is applied to take the shear interaction into account and to obtain the applied bending stiffness in the stick domain EI_{stick}:

$$EI_{stick} = \frac{EI_{plane}}{1 + \frac{EAsin^2\alpha}{kR^2}} \tag{2.6}$$

where the shear stiffness parameter k is calculated from the below input as:

$$k = G\frac{b}{t_{eff}} \tag{2.7}$$

where G is the tape layer shear modulus and t_{eff} is the effective tape thickness and b is the width of the tensile wire. It is to be noted that this functionality should be used with care, as a too small stiffness will yield unconservative stresses. Therefore these parameters should be established based on measured moment-curvature data.

For the sandwich beam model approach, the shear interaction stiffness needed to maintain plane surfaces remain plane, k_{plane} is firstly established.

$$k_{plane} = 12EA\frac{sin^2\alpha}{R^2} \tag{2.8}$$

and then k_{stick} is established from the shear interaction stiffness k defined above:

$$\frac{1}{k_{stick}} = \frac{1}{k_{plane}} + \frac{1}{k} \tag{2.9}$$

Note that one exception is made for ITCODE31 and ITCODE32. For these options the user can still actively select the old friction model, and also include additional shear interaction. However, this option should be used with care. Also note that for the remaining modelling options for FLEXCROSS and 353FLEXCROSS the activation of the old friction model will deactivate the shear stiffness input given.

ELPROP ELGRP SHEARMODEL *TEFF GMOD* [*IFRICTION KSURF KYEL*]

TEFF: The effective thickness of the tape layer. (unit: L)

GMOD: Shear modulus of the tape layer. If no values are given for the TEFF and GMOD by using D for each parameter, the default plane surface remain plane model will be applied. This is only relevant when one wants to activate the new friction model assuming no softening from the tape layers (unit: FL^{-2}).

IFRICTION: A non-zero value will activate the new friction model that is based on a constant stick stiffness, while zero will activate the old friction model. Optional. Default value: 1

KSURF: A user defined surface stiffness that will overrule the value that is automatically calculated by applying the 353FLEXCROSS procedure. Optional. Default value: 0 (then the value obtained by the 353FLEXCROSS procedure will be used) (unit: FL^{-2}).

KYEL: A user defined lateral transverse elastic stiffness. By activating the new friction model (IFRICTION) in combination with the KYEL parameter, the friction model will be 1D longitudinal, whereas an elastic spring with stiffness KYEL will

be introduced in the lateral transverse direction. Optional. Default value: 0. (unit: FL^{-2}).

2.20.15 SOILCONTACT

ELPROP ELGRP SOILCONTACT TX TY TZ TTX THISTX THISTY THISTZ THISTTX XNAME YNAME ZNAME TXNAME [T0_UZ THISTUZ UZNAME]

TX: Activation time for spring in pipe axial direction.

TY: Activation time for spring in pipe transverse direction.

TZ: Activation time for spring in pipe-seabed normal direction.

TTX: Activation time for spring about pipe axial rotation DOF.

THISTX: Time history for scaling of spring force and stiffness in pipe axial direction.

THISTY: Time history for scaling of spring force and stiffness in pipe transverse direction.

THISTZ: Time history for scaling of spring force and stiffness in pipe-seabed normal direction.

THISTTX: Time history for scaling of spring moment and stiffness about pipe axial rotation DOF.

XNAME: Material name for spring in pipe axial direction. Force unit: F/L. Displacement unit: L.

YNAME: Material name for spring in pipe transverse direction. Force unit: F/L. Displacement unit: L.

ZNAME: Material name for spring in pipe-seabed normal direction. Force unit: F/L. Displacement unit: L.

TXNAME: Material name for spring about pipe axial rotation DOF. Moment unit: F. Displacement unit: rad.

T0_UZ: Reference time for the KP-based soil embedment. Optional.

THISTUZ: Time history for scaling of KP-based soil embedment. Optional.

UZNAME: Table name for KP-based soil embedment. Optional.

The main purpose of the time-controlled soil springs is to enhance the modelling capabilities of the analysis start-up phase. A typical application is to avoid singular stiffness matrix in static analysis due to e.g. lack of seabed contact or lack of axial and transverse contact element stiffness. All material curves must be defined by the **HYCURVE** material type in Section 2.26.5. The springs are active regardless of whether contact is present or not. The springs are deactivated by using zero load factor in the time history definitions. Spring forces and moments are added to the ordinary soil contact

forces and moments, and will be included in results reported by BFLEX2010POST. The element property type is optional. No time-controlled springs will be present if the **ELPROP** input card for **SOILCONTACT** is omitted.

KP-based soil embedment is modeled by the 2-dimensional table **UZNAME**. The first table column contains the KP-values with coordinate reference as defined by the **COSURFPR** card in Section 2.8. The second column of the table contains the soil embedment $\bar{u}_{z0}$. See Section 2.34 for definition of the table input format.

The reference time **T0_UZ** refers to the simulation time where the pipeline shall have soil penetration equal to the specified KP-based soil embedment $\bar{u}_{z0}$. To achieve this, BFLEX2010 adjusts for the contact element penetration u_z as follows,

$$u_{z0} = \begin{cases} f_1 \cdot \bar{u}_{z0} - f_2 \cdot u_z(t_0) & \text{if} \quad u_z(t_0) \le 0.0 \quad , \quad t > t_0 \quad \text{(contact at } t{=}t_0\text{)} \\ f_1 \cdot \bar{u}_{z0} & \text{if} \quad u_z(t_0) \ge 0.0 \quad , \quad t > t_0 \quad \text{(no contact at } t{=}t_0\text{)} \\ f_1 \cdot \bar{u}_{z0} - f_2 \cdot u_z(t) & \text{if} \quad u_z(t) \le 0.0 \quad , \quad t \le t_0 \end{cases} \tag{2.10}$$

where t_0=**T0_UZ**, the load factor f_1 is taken from the **THISTUZ** load history, f_2 is set equal to f_1 but limited by $f_2 \le 1.0$, u_{z0} is the stressfree initial displacement that will be applied for the contact element, $\bar{u}_{z0}$ is the specified KP-based soil embedment at simulation time t_0=**T0_UZ**, and u_z is the contact element penetration used for computing the normal contact force F_z.

The sign convention $u_z < 0.0$ means that the pipeline penetrates a stress-producing distance into the seabed, and $u_{z0} < 0.0$ means that the pipeline sinks a stressfree distance into the seabed. The quantity u_{z0} represents the irreversible modification of the virgin seabed, which when added together with the contact element penetration u_z gives the total soil embedment $\bar{u}_{z0}$ at simulation time t_0=**T0_UZ**.

EXAMPLE:

```
#       elgrp    type          tx   ty   tz   trx   thistx  thisty  thistz  thistrx
ELPROP  seabed   soilcontact   0.0  0.0  0.0  0.0   400     400     400     400
#
# xname   yname    zname    txname    t0_uz  thistuz  uzname
soilxh    soilyh   soilzh   soiltxh   1.0     450     uzini
#
#      name   ncol
TABLE  uzini  2
-100.0  -0.2
 100.0  -0.1
 400.0  -0.2
```

2.21 ENVRES_ - Envelope results

By the **ENVRES** cards, user select results for a given range of element or nodal nodes will be stored on the .raf file on a format that enable envelope results to be presented on ASCII matrix plot format by the BFLEX2010POST postprocessor. The following format is applied:

```
ENVRES_N, E OR I ...
```

where **E** means element results, **N** means nodal results and **I** means Gaussian integration station results. The allowable options are defined below:

2.21.1 ENVRES_N

For **ENVRES** with option **_N** (nodal results) the following format applies:

```
ENVRES_N TYPE NODE1 NODE2 DOF STAT [START STOP]
```

TYPE: Type or result, where allowable values are:

- **1 :** nodal displacement
- **2 :** nodal velocity
- **3 :** nodal acceleration
- **4 :** relative displacement

NODE1: First node ID

NODE2: Last node ID

DOF: Degree of freedom number;

- **1 :** x-direction
- **2 :** y-direction
- **3 :** z-direction
- **4 :** rotation x-direction
- **5 :** rotation y-direction
- **6 :** rotation z-direction

TIME0: Time to store static information.

START: Start time for storing results, optional.

STOP: End time for storing results, optional.

2.21.2 ENVRES_E

For elements of type PIPE, CONT and SPRING, the **ENVRES** card can be used with the **_E** option (element results). The following format applies:

ENVRES_E TYPE EL1 EL2 ELNOD DOF TIME0 [START STOP]

TYPE: Type of result, where allowable values are:

1 : = displacement
2 : = forces and moments
3 : = torsion and curvatures
4 : = absolute curvature

EL1: Element 1 ID.

EL2: Element 2 ID. Results are stored for element **EL1** to **EL2**.

ELNODE: Element node, i.e. element end. (1 or 2 for PIPE, only 1 for CONT).

DOF: Degree of freedom number. The meaning is element type dependent:

PIPE31-39, SPRING136-137 : :

1 : = x-disp/x-force /x-torsion
2 : = y-disp/y-force/y-curvature
3 : = z-disp/z-force/z-curvature
4 : = x-rotation/x-moment
5 : = y-rotation/y-moment
6 : = z-rotation/z-moment

CONT : contact element types:

1 : = x-disp, x-force
2 : = y-disp, y-force
3 : = z-disp, z-force

TIME0: Time to store static information.

START: Start time for storing results, optional.

STOP: End time for storing results, optional.

2.21.3 ENVRES_I

For elements of type PIPE33, the **ENVRES** card can be used with the **_I** option (Gaussian interpolation station results). The following format applies:

ENVRES_I TYPE EL1 EL2 IGAU IPOINT TIME0 [START STOP]

TYPE: Type of result, where allowable values are:

1 : = sigma-xx

2 : = strain-xx

EL1: Element 1 ID.

EL2: Element 2 ID. Results are stored for element **EL1** to **EL2**.

IGAU: Element integration station 1-3, 1 and 3 at element ends.

IPOINT: Integration point number, max NPOINT see Section 2.6.

TIME0: Time to store static information.

START: Start time for storing results, optional.

STOP: End time for storing results, optional.

EXAMPLE:

```
#
#          type  nod1  nod2  dof statstep
ENVRES_N   1     1     441   1   1
ENVRES_N   1     1     441   3   1
#
#          type  el1   el2   end  dof statstep
ENVRES_E   2     2301 2316   1    3   1
ENVRES_E   2     1001 1200   1    3   1
ENVRES_E   2        1  440   2    1   1
ENVRES_E   2        1  440   2    5   1
ENVRES_E   3        1  440   1    2   1
#
#          strain   el1  el2  gauss  point   statstep
ENVRES_I   2        341  440  1      4       1
```

2.22 FATPROP - FATigue properties

The **FATPROP** card enables the user to associate fatigue properties stored on specific files names to the different material names

The format is as follows:

FATPROP NAME FILE

where

NAME: Material name.

FILE: File name at which the fatigue data are stored.

2.22.1 Fatigue file format

One line on the format:

NFDPO R1 IGERB INTCO SCF SIGUTS

NFDPO : The number of points in the fatigue S-N diagram

R : The R-ratio defined as $\sigma_{min}/\sigma_{max}$ for the S-N-diagram

IGERB : Method for taking the mean stress into account

- **0 :** No mean stress is taken into account
- **1 :** Goodman interpolation mean stress calculated as $\sigma_{xx} + \sigma_{yy} + \sigma_{zz}$
- **2 :** Gerber interpolation mean stress calculated as $\sigma_{xx} + \sigma_{yy} + \sigma_{zz}$
- **3 :** Goodman interpolation mean stress calculated as $\bar{\sigma} = \sqrt{\sigma_{xx}^2 + \sigma_{yy}^2 + \sigma_{zz}^2 - \sigma_{xx}\sigma_{yy} - \sigma_{zz}\sigma_{yy} - \sigma_{xx}\sigma_{zz} + 3\sigma_{xy}^2 + 3\sigma_{zy}^2 + 3\sigma_{xz}^2}$
- **4 :** Gerber interpolation mean stress calculated as $\bar{\sigma} = \sqrt{\sigma_{xx}^2 + \sigma_{yy}^2 + \sigma_{zz}^2 - \sigma_{xx}\sigma_{yy} - \sigma_{zz}\sigma_{yy} - \sigma_{xx}\sigma_{zz} + 3\sigma_{xy}^2 + 3\sigma_{zy}^2 + 3\sigma_{xz}^2}$

INTCO : Method for interpolation in S-N diagram:

- **1 :** both stress and N in log scale
- **2 :** stress in linear scale, N in log scale

SCF : Stress concentration factor (unit:)

SIGUTS : Ultimate stress for mean stress correction (unit: F/L^2)

Then a number of NFDPO lines on the format:

POINT SRANGE NCYFAL

POINT : The point number =1- .... Number of points.

SRANGE : Stress range (in increasing order), starting with the minimum threshold value (unit: F/L^2)

N : Corresponding number of cycles to failure.

Note: Only numeric values in the fatigue data file. No lines starting with # is possible in the fatigue data file, i.e. just numbers

2.23 GEOM - geometry of body

The **GEOM** card can be applied to specify a geometry for an element of type BODY. The geometry will define the contact surface between a pipe and the body if contact elements are defined between them. Note that only CONT152 and CONT153 can be applied for this sort of contact.

A geometry is defined by a sequence of segments, where each segment has its own geometric shape. The orientation of the geometry is the same as the element it is connected to. The element node will be in center of the first segment, unless eccentricities are given by the **ELECC** card, see Section 2.15. Note that the same eccentricities must be defined for the contact elements as well.

The format of the card is as follows:

```
GEOM NAME NSEG TYPE THETA NVIS L ....
                    .... ....
```

where

NAME: Name of geometry. This will be used as reference in the **ELPROP** card, see Section 2.20.

NSEG: Number of segments in the geometry. A geometry can be built from several segment types aligned after each other in the sequence they are given in the geometry card.

TYPE: Type of geometry for the segment. The following types are implemented:

BELLMOUTH : Bellmouth.

CYLINDER : Cylinder.

For **TYPE** = **BELLMOUTH** the format is:

```
GEOM NAME NSEG BELLMOUTH THETA NVIS LEN DIAM RADSTART RADEND PHI0
                    .... ....
```

where:

THETA: Rotation about local x-axis. This rotation is relative to the previous segment. For the first segment, the rotation is relative to the orientation of the body element.

NVIS: Number of visual sections.

LEN: Length of the segment. (unit: L)

DIAM: Smallest diameter of segment. (unit: L)

RADSTART: Curvature radius at start of segment. (unit: L)

RADEND: Rate of curvature radius change along segment so that 1/RAD=1/RADSTART+1/RADEND*S where s is curvilinear length along segment (unit: L^2)

PHI0: Slope at start of segment. The slope is measured along the longitudinal direction, and relative to the center line of the bellmouth.

The curvature of the bellmouth is taken to be zero if the end curvature and the start curvature is estimated to differ by less than a hundredth of the start curvature.

EXAMPLE:

```
# Geometry data
#
#    name    nseq          type  theta   nvis    len     diam  radstart radend    phi0
GEOM geo1       2     bellmouth    0.0     10    2.0   0.1650   150.0     0.0     0.0000
                      bellmouth    0.0     10    2.0   0.1917    20.0     0.0     0.0133
```

For **TYPE** = **CYLINDER** the format is:

GEOM NAME NSEG CYLINDER THETA NVIS LEN DIAM1 DIAM2
....

where:

THETA: Rotation about local x-axis. This rotation is relative to the previous segment. For the first segment, the rotation is relative to the orientation of the body element.

NVIS: Number of visual sections.

LEN: Length of the segment.

DIAM1: Diameter of first segment.

DIAM2: Diameter of last segment.

EXAMPLE:

```
#     name    nseg    type      theta  nvis  len   diam1   diam2
GEOM  cyli2    1     CYLINDER   0.0    4     3.5   0.17    0.17
```

2.24 HEAD - HEADing

The **HEAD** command is used to supply a descriptive text to the analysis. Up to six lines of text may be used to describe the analysis. Each line is started with **HEAD** followed by an arbitrary text string. The format is as follows:

HEAD TEXT

where:

TEXT: Descriptive analysis text.

EXAMPLE:

```
#
HEAD This is a test example
HEAD Elastic material model - 440 pipe elements
#
```

2.25 INISTR - initial strain loading

The initial element strain loading is applied by the following format:

INISTR HIST DIR IEL1 ISTR1 [IEL2 ISTR2] **[REPEAT N ELINC]**

where

HIST: Load history number.

DIR: Element load direction relative to the element axis (**1** = axial strain , **4-6** = torsion and curvatures about x-z).

IEL1: First element ID.

ISTR1: Initial strain for the first element.

IEL2: Last element ID, optional.

ISTR2: Initial strain for the last element, optional. Linear interpolation is applied for intermediate elements.

If the **REPEAT** command is used then the previous sequence of load generation is repeated:

N: Number of repetitions.

ELINC: ELement increment.

N times with element increment m. To summarise:

DIR has value	**ISTR1/ISTR2** is
1	axial strain along local x axis (unit: -)
4	the torsion about local x axis (unit: L^{-1})
5	curvature about local y axis (unit: L^{-1})
6	curvature about local z axis (unit: L^{-1})

For the HELIX elements if **DIR**=**-4**, **-5** or **-6**, i.e. given as a negative number, initial strains are automatically calculated as the torsion, normal curvature and transverse curvatures being a result of the helical path defined in the **ELPROP** card, see Section 2.20.

EXAMPLE:

```
#       hist  dir  iel1  torsion
#
INISTR  200   4    100   20.0
                   103   20.0
#      n m
REPEAT 4 4
```

2.26 MATERIAL - MATERIAL properties

The following material types are available in version 3.3.1 of the program:

LINEAR : Linear material properties for the elastic pipe elements. Relevant for PIPE31, CABLE111 and HELIX231.

ELASTOPLASTIC : Elastoplastic strain-stress material behaviour with kinematic/isotropic hardening due to the elastoplastic pipe elements. Relevant for PIPE33 and HELIX233

EPCURVE : Elastoplastic material behaviour with kinematic/isotropic hardening.

HYCURVE : Hyperelastic (non-linear elastic) material behaviour.

FRICTION : Various friction models with user defined elastic stiffness.

SEA : Sea properties, for the SEA150 element type only.

RESULTANT : User defined material property based on describing the material curves in each direction for the COMPIPE42 element type.

R_CONTACT : User defined material property based on describing the material curves in the x-, y- and z-directions and the x-rotation for the seabed contact element type CONT126. The purpose of the model is to enable modelling of x-moment induced by pure x-rotation with zero contribution from the y-displacement. As

default, the x- and y-direction material curves are to be specified as displacements versus unit force curves. By specifying **USERDEFINED** at the end of the contact card, the x- and y-curves are to be specified as displacement versus force. Then the x- and y-forces will be independent of the z-force. The element length is taken into account so the material curves are per unit length.

CONTACT : User defined material surface property based on describing the material curve in the x-, y- and z- directions for the contact element types. As default, the force x and y components are multiplied with the z-force reaction and a friction coefficient. This means that the x- and y-material curves are to be specified as displacements versus unit force curves. By specifying **USERDEFINED** at the end of the card, the x- and y- curves are to be specified as displacement versus force curves. Then the x- and y-forces will be independent of the z-force. Using this functionality together with HCONT453 OR 463 the gap locic will further be turned off. Note also that for these element types only hycurve is allowed for in the contact z-direction, whereas epcurve (plastic) curves can be applied to model unbonded or bitumen effects in local x- and y- directions. For element type 125, 126, 130 152, 453, 454, 463 and 473 the element length is taken into account, i.e. the diagrams is given per element length.

ISOCONTACT : Isotropic contact material for element types 124, 130, 152 and 164.

ISOKXYCONTACT : Isotropic contact material with constant stick stiffness for element type 164.

ELASTIC : Elastic material behaviour (linear elastic).

HYPERELASTIC : Hyperelastic strain-stress material behaviour (non-linear elastic).

GENSPRING : User defined material property based on describing the material curve in the x and y and z- directions for both force and moment in 6 DOFs. This option is used for the element type SPRING137. The element may be activated at an arbitrary load step in the different directions, see Section 2.26.16.

HELSPRING : User defined material property based on describing the material friction unit-curve in the local curvilinear x-direction for the HELSPR437 element type. The force is automatically multiplied with the radial pressure from the helix element attached to node 2 in the helix element definition, see Section 2.13, and the friction coefficient defined here. The friction spring is activated at an arbitrary load step, see Section 2.20.

FRICONTACT : The contact is treated based on using a fixed penalty parameter value. Several friction models are included, (Dai et al., 2017):

- An isotropic static model including static and dynamic friction coefficients as well as user defined elastic stiffness and transitione stiffness from static friction to dynamic friction.

- An isotropic dynamic model including static and dynamic friction coefficients, user defined elastic stiffness and a dynamic model describing the transition from static friction to dynamic friction (based on the velocity).
- Anisotropic versions of both the above models

The model is applicable for the HCONT453, HCONT454, HCONT463, , HCONT464 and HCONT473 elements.

FRICVISC : The contact is treated as a combination of Coulomb friction and viscoelastic behaviour. The model is applicable for the HCONT463 and HCONT464 elements.

Some of the parameters may be dummy, as the input is organized in the same manner for different materials. The user should specify a value of correct type, but the specified value will not be applied.

The format is as follows:

```
MATERIAL MNAME MTYPE ...
```

where

MNAME: Material name.

MTYPE: Material type as defined above.

The following parameters depend on the material type as specified in the following.

2.26.1 LINEAR (PIPE31, HELIX231, 234, FLEXCROSS, 353FLECROSS, PIPE31, HSHEAR353, 363, 364)

```
MATERIAL MNAME LINEAR POISS TALFA TECOND HEATC BETA EA EIY EIZ
GIT EM GM [DENSITY ETRANS]
```

POISS: Poisson's ratio (-)

TALFA: Temperature elongation coeff. (unit K^{-1})

TECOND: Thermal conductivity (dummy) (unit: W/mC^o)

HEATC: Heat capacity (dummy) (unit: unit: J/kgC^o)

BETA: Tension/torsion coupling parameter. Normally zero, but can be specified to capture torsion effect if the relation between tension/torsion is known. (unit: L)

EA: Axial stiffness (unit: F)

EIY: Bending stiffness about y axis (unit: FL^2)

EIZ: Bending stiffness about z axis (unit: FL^2)

GIT: Torsion stiffness (unit: FL^2)

EM: Young's modulus (unit: FL^{-2})

GM: Shear modulus (unit: FL^{-2})

DENSITY: Density (unit: ML^{-3})

ETRANS: Transvere Young's modulus (unit: FL^{-2})

EXAMPLE:

```
#        name     type    pois talfa  tecond heatc beta ea    eiy    eiz  git   em  gm
MATERIAL pipemat1 linear 0.3  1.1e-5 50     800   0    1.4e4 7.6e2 7.6e2 5.8e2 2e5 8e4
```

2.26.2 ELASTIC (FLEXCROSS, 353FLEXCROSS, HSHEAR353, 363, 364)

MATERIAL MNAME ELASTIC POISS DENSITY TALFA TECOND HEATC EM

POISS: Poisson's ratio (-)

DENSITY: Density (unit: ML^{-3})

TALFA: Temperature elongation coeff. (unit K^{-1})

TECOND: Thermal conductivity (dummy) (unit: W/mC^o)

HEATC: Heat capacity (dummy) (unit: unit: J/kgC^o)

EM: Young's modulus (unit: FL^{-2})

EXAMPLE:

```
#        name     type    pois density talfa  tecond heatc   em
MATERIAL pipemat1 elastic 0.3  7850 1.1e-5 50     800   0   2e5
```

2.26.3 ELASTOPLASTIC (FLEXCROSS, PIPE33, HELIX233)

MATERIAL MNAME ELASTOPLASTIC IHARD POISS RHO0 TALFA TECOND

HEATC EPS SIGMA

... ...

IHARD: Hardening parameter 0.0-1.0 or 2.0-3.0; 0.0 = isotropic hardening; 1.0 = kinematic hardening with accumulated plsatic strains. 2.0 and 3.0 is isotropic and kinematic hardening respectively, but with no accumulation of plastic strains

for reverse axial loading. Value can also be between 2.0-3.0, corresponding to values 0.0-1.0 in terms of hardening.

POISS: Poisson's ratio (-)

RHO0: Density (dummy) (unit: ML^{-3})

TALFA: Temperature elongation coeff. (unit K^{-1})

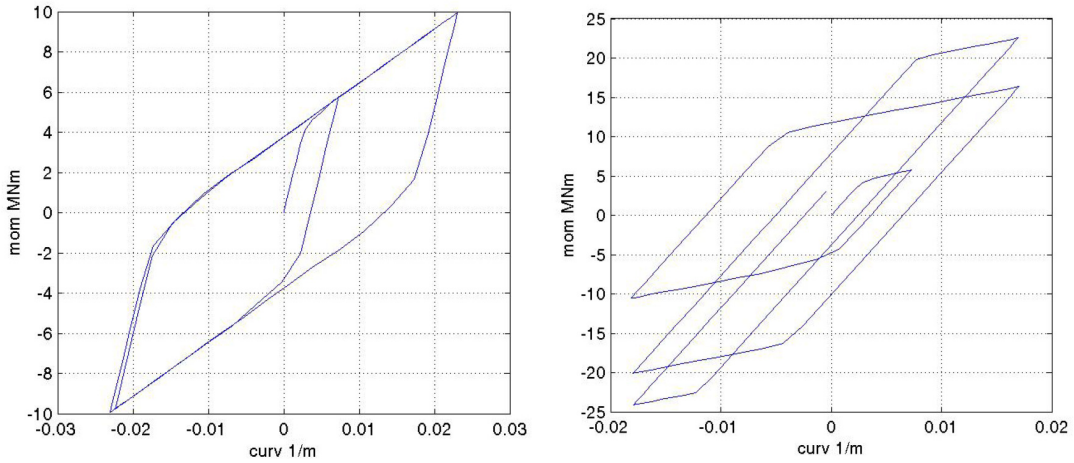
TECOND: Thermal conductivity (dummy) (unit: $EL^{-1}TE^{-1}$)

HEATC: Heat capacity (dummy) (unit: $E^{-1}M^{-1}TE^{-1}$)

EPS: Strain always starting at 0

SIGMA: Stress (unit: FL^{-2}) always starting with 0 and then only positive paired values of strain and stress.

Any number of pairs of **EPS** and **SIGMA** can be given, and linear interpolation is applied between the supplied data. The first non-zero strain-stress pair is taken as the proportionality limit. The hardening parameter (**IHARD**) controls the behaviour of the yield surface. Kinematic hardening (i.e. elastic range is constant equal to $2\sigma_y$) is specified by a hardening parameter equal to 1.0. Isotropic hardening (i.e. elastic range is extended as function of strain experienced by the material) is specified by a hardening parameter equal to 0.0. An illustration for a simply supported beam exposed to repeated rotations in one end is shown below in Fig. 2.29. Note that the first rotation has smaller amplitude than the following.



(a) Kinematic hardening (ihard= 1.0) (b) Isotropic hardening (ihard=0.0)

Figure 2.29: Illustration of material hardening effect.

EXAMPLE:

```
#
#        name     type          ihard poiss  ro   talfa   tecond heatc
MATERIAL pipemat2 elastoplastic 1.0     0.3    7850 1.17e-5 50     800
#
#        eps         sigma ..1,n
```

```
0           0
1.691E-03   3.50E+02
2.000E-03   3.90E+02
2.500E-03   4.15E+02
3.000E-03   4.27E+02
3.500E-03   4.35E+02
4.000E-03   4.41E+02
5.010E-03   4.50E+02
6.000E-03   4.57E+02
```

2.26.4 EPCURVE

```
MATERIAL MNAME EPCURVE IHARD EPS SIGMA
                                ...   ...
```

IHARD: Hardening parameter; **0** = isotropic hardening; **1** = kinematic hardening.

EPS: Strain, displacement, torsion or curvature depending on which quantity to be described, starting at 0.

SIGMA: Stress, unit resultant or resultant values depending on which quantity to be described. The material curve should always start at 0 and then contain only positive paired values.

Between specified points, linear interpolation is applied. Values outside the specified range are extrapolated.

Table 2.2 gives an overview of which dimensions to apply for **EPS** and **SIGMA**, depending on the setting in which the **EPCURVE** is applied.

Table 2.2: Units for **EPS** and **SIGMA**.

Element type	Material type	Option	DOF	EPS-unit	SIGMA-unit
COMPIPE42	RESULTANT		axial	- (strain)	F (force)
	RESULTANT		torsion	L^{-1} (torsion)	FL (torsion moment)
	RESULTANT		bending	L^{-1} (curvature)	FL (bending moment)
CABLE111			axial	- (strain)	F (force)
CONT124	CONTACT ISOCONTACT		z-dir	L (displacement)	F (force)
	CONTACT	COULOMB (default)	x- and y-dir	L (displacement)	- (unit force)
	CONTACT	USERDEFINED	x- and y-dir	L (displacement)	F (force)
	ISOCONTACT		x- and y-dir	L (displacement)	- (unit force)
SZHELIX HELIX HELSPR	CONTACT		z-dir	L (displacement)	FL^{-1} (force pr.length)

Continued on next page

Element type	Material type	Option	DOF	EPS-unit	SIGMA-unit
	CONTACT	COULOMB (default)	x- and y-dir	L (displacement)	L^{-1} (unit force pr.length)
	CONTACT	USERDEFINED	x- and y-dir	L (displacement)	FL^{-1} (force pr.length)
CONT126	CONTACT R_CONTACT		z-dir	L (displacement)	FL^{-1} (force pr. length)
	CONTACT R_CONTACT	COULOMB (default)	x- and y-dir	L (displacement)	L^{-1} (unit force pr. length)
	CONTACT R_CONTACT	USERDEFINED	x- and y-dir	L (displacement)	FL^{-1} (force pr. length)
	R_CONTACT	COULOMB (default)	x-rot	Rad (rotation)	L^{-1} (unit y-force pr. length)
	R_CONTACT	USERDEFINED	x-rot	Rad (rotation)	F (force pr. length)
CONT130	CONTACT ISOCONTACT ISOKXYCONTACT		z-dir	L (displacement)	FL^{-1} (force pr. length)
	CONTACT ISOCONTACT ISOKXYCONTACT	COULOMB (default)	x- and y-dir	L (displacement)	L^{-1} (unit force pr. length)
	CONTACT	USERDEFINED	x- and y-dir	L (displacement)	FL^{-1} (force pr. length)
SPRING137	GENSPRING		x-, y, and z-dir	L (displacement)	F (force)
	GENSPRING		x-, y- and z-rot	Rad (rotation)	FL (moment)
CONT152	CONTACT ISOCONTACT ISOKXYCONTACT		z-dir	L (displacement)	FL^{-1} (force pr. length)
	CONTACT ISOCONTACT ISOKXYCONTACT	COULOMB (default)	x- and y-dir	L (displacement)	L^{-1} (unit force pr. length)
	CONTACT	USERDEFINED	x- and y-dir	L (displacement)	FL^{-1} (force pr. length)
CONT164	CONTACT ISOCONTACT ISOKXYCONTACT		z-dir	L (displacement)	F (force)
	CONTACT ISOCONTACT ISOKXYCONTACT	COULOMB (default)	x- and y-dir	L (displacement)	- (unit force)
	CONTACT	USERDEFINED	x- and y-dir	L (displacement)	F (force)
HCONT453	CONTACT ISOCONTACT		z-dir	L (displacement)	F (force pr. length)
	ISOCONTACT		x- and y-dir	L (displacement)	- (unit force)
	CONTACT	Default	x- and y-dir	L (displacement)	- (unit force)
	CONTACT	USERDEFINED	x- and y-dir	L (displacement)	F (force pr. length)
HCONT454	CONTACT ISOCONTACT		z-dir	L (displacement)	F (force pr. length)
	ISOCONTACT		x- and y-dir	L (displacement)	- (unit force)
	CONTACT	Default	x- and y-dir	L (displacement)	- (unit force)

Continued on next page

Element type	Material type	Option	DOF	EPS-unit	SIGMA-unit
	CONTACT	USERDEFINED	x- and y-dir	L (displacement)	F (force pr. length)
HCONT463	CONTACT ISOCONTACT		z-dir	L (displacement)	F (force pr. length)
	ISOCONTACT		x- and y-dir	L (displacement)	- (unit force)
	CONTACT	Default	x- and y-dir	L (displacement)	- (unit force)
	CONTACT	USERDEFINED	x- and y-dir	L (displacement)	F (force pr. length)
HCONT473	CONTACT ISOCONTACT		z-dir	L (displacement)	F (force pr. length)
	ISOCONTACT		x- and y-dir	L (displacement)	- (unit force)
	CONTACT	Default	x- and y-dir	L (displacement)	- (unit force)
	CONTACT	USERDEFINED	x- and y-dir	L (displacement)	F (force pr. length)

EXAMPLE:

```
#        name  type     ihard  eps    sig
MATERIAL soilx epcurve    1    0.000  0.0
                               0.005  1.0
                               1.000  1.0
```

2.26.5 HYCURVE

MATERIAL MNAME HYCURVE EPS SIGMA
... ...

EPS: Strain, displacement, torsion or curvature depending on which quantity to be described

SIGMA: Stress or resultant values depending on which quantity to be described

Between specified points, linear interpolation is applied. Values outside the specified range are extrapolated.

To help in convergence of the CONT152 element, a smooth stiffness transition has been implemented. The stiffness goes from zero to the full value over a length which stretches 0.001 times the length from the origin to the **EPS**-coordinate of the first material point on the negative side. This modification only affects the stiffness, and

not the force. Consequently, the final equilibrium state will not be influenced by the stiffness modification. For CONT152, the first material point on the negative side is suggested to be less than 1 percent of the last material point as can be seen in the example below.

Refer to Table 2.2 for an overview of which dimensions to apply for **EPS** and **SIGMA**, depending on the setting in which the **HYCURVE** is applied.

EXAMPLE:

```
#         name    type      eps         sig
MATERIAL  soilx   hycurve  -1000.000   -10.0
                            1000.000    10.0
#         name    type           eps       sig
MATERIAL  bm_cont_z  hycurve  -1000.0   -100000.0
                                -10.0     -1000.0
                                  0.0         0.0
                               1000.0    100000.0
```

2.26.6 ISOHYCURVE

MATERIAL MNAME ISOHYCURVE EPS SIGMA
... ...

EPS: Strain, displacement, torsion or curvature depending on which quantity to be described

SIGMA: Stress or resultant values depending on which quantity to be described

ISOHYCURVE is used for applications which need symmetry about origo. The first eps sigma pair must thus be in origo.

Refer to Table 2.2 for an overview of which dimensions to apply for **EPS** and **SIGMA**, depending on the setting in which the **ISOHYCURVE** is applied.

EXAMPLE:

```
#         name    type         eps         sig
MATERIAL  soilx   isohycurve     0.0        0.0
                              1000.000    10.0
```

2.26.7 SEA

MATERIAL MNAME MTYPE SEADEN

SEADEN: Sea density (unit: ML^{-3})

EXAMPLE:

```
#        name      type    density
MATERIAL seamat    sea     1000e-6
```

2.26.8 RESULTANT (COMPIPE42, HELIX235)

MATERIAL MNAME RESULTANT POISS RHO TALFA TECOND HEATC BETA FQX FKX FKY FKZ FF REFP IPDER EPDER REFT TDER AXNAME TORNAME BENDNAME

POISS: Effective Poisson's ratio ν (-) based on $\varepsilon = (1-2\nu)\frac{\pi}{4}(p_i D_i^2 - p_0^2)$

RHO: Density (Dummy) (unit: ML^{-3})

TALFA: Temperature elongation coeff. α (-) $\varepsilon = \alpha \Delta T$

TECOND: Thermal conductivity (Dummy) (unit: $\mathrm{EL}^{-1}\mathrm{TE}^{-1}$)

HEATC: Heat capacity (Dummy) (unit: $\mathrm{E}^{-1}\mathrm{M}^{-1}\mathrm{TE}^{-1}$)

BETA: Tension/torsion coupling parameter β. For a tensile armour of a flexible pipe with lay angle α and radius R, the strain due to centreline strain and torsion is: $\varepsilon_x = u_{x,x}\cos^2\alpha + R\sin\alpha\cos^2\alpha\theta_{x,x} = u_{x,x}\cos^2\alpha + \beta\theta_{x,x}$(unit:L)

FQX: Axial stress factor f_{qx} (-) Axial stress = Axial force × factor

FKX: Torsion stress factor f_{kx} (-). For element type COMPIPE42: Shear stress = Torsion × factor. For element type HELIX235 Shear stress = Torsion moment × factor.

FKY: y-curvature elastic stress factor f_{ky} (-). For element type COMPIPE42: Bending stress = y-curvature × factor. For element type HELIX235: Bending stress = y-moment × factor.

FKZ: z-curvature elastic stress factor f_{kz} (-). For element type COMPIPE42: Bending stress = z-curvature × factor. For element type HELIX235: Bending stress = z-moment × factor.

FF: friction stress conversion factor (-) Bending stress = Bending moment × factor

REFP: Reference pressure (Dummy) (unit: FL^{-2}).

IPDER: Friction moment internal pressure rating factor (Dummy) referring to Section 2.32 and see Section 2.35. The parameter can be used to increase the friction

moment from the value given on the material curve defined at REFP by linear internal pressure scaling (unit: L^3).

EPDER: Friction moment external pressure rating factor (Dummy) referring to Section 2.31 and see Section 2.35. The parameter can be used to increase the friction moment from the value given on the material curve defined at zero external pressure by linear internal pressure scaling (unit: L^3).

REFT: Reference temperature (Dummy) (unit: C).

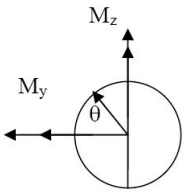
TDER: Temperature derating factor (Dummy) referring to Section 2.37 and Section 2.35 The parameter can be used to decrease the bending stiffness beyond the friction moment point from the value given on the material curve defined at REFT by linear temperature scaling(unit: C^{-1}).

AXNAME: Material name in axial direction (epcurve or hycurve)

TORNAME: Material name in torsion (epcurve or hycurve)

BENDNAME: Material name in bending (epcurve, hycurve (only HELIX235) or isohycurve (only COMPIPE42))

Note:



$\sigma_{xx} = F_x f_{qx} + \kappa_y f_{ky} + \kappa_z f_{kz} - M_y f_f \sin\theta + M_z f_f \cos\theta$
and
$\sigma_{xy} = \kappa_x f_{kx}$

EXAMPLE:

```
#-------------------------------------------------------------------------------
#        name      type      poiss density Thermexp Cond Heatc Beta Axs    Tors
MATERIAL restrmat1 resultant 0.3   0       1.17e-5  50   800   0.0  2.07e5 27152
#
# ycurs zcurs momf axnam tornam bendnam
  7025  7025  81   axmat tomat  bendmat
```

It is noted that the material curves need to be defined using the **HYCURVE**, **ISOHYCURVE** or **EPCURVE** options. For both cases the curve is to be defined as a consecutive number of points defining strain/curvature versus force/moment.

2.26.9 CONTACT (CONT126, 130, 152, HELIX231, 233, 234, 235))

```
MATERIAL MNAME CONTACT MUX MUY XNAME YNAME ZNAME
[ [XYCHOICE] [ NX NY PX_1 PXNAME_1 ... PX_NX PXNAME_NX PY_1
PYNAME_1 ... PY_NY PYNAME_NY ] ]
```

MUX: Friction coefficient in local x-direction (unit: -).

MUY: Friction coefficient in local y-direction. Negative value will provide isotropic friction based on **MUX** for element type CONT126 (unit: -).

XNAME: Local x-direction material curve name.

YNAME: Local y-direction material curve name.

ZNAME: Local z-direction material curve name.

XYCHOICE: Enables overruling of the default Coulomb friction model with userdefined characteristics for the x- and y-directions. Optional. Allowable values:

COULOMB : The **XNAME** and **PXNAME_I** curves are scaled by the product of the local z-direction force and **MUX**, while the **YNAME** and **PYNAME_I** curves are scaled by the product of the local z-direction force and **MUY** This is the default behaviour.

USERDEFINED : The **XNAME**, **YNAME**, **PXNAME_I** and **PYNAME_I** curves as defined by the user are applied.

COULOMB-COULOMB-COULOMB : Scaling according to **COULOMB** is applied for all material curves in the local x- and y-directions. Equivalent to specifying **COULOMB** only.

COULOMB-USERDEFINED-USERDEFINED : Scaling according to **COULOMB** for **XNAME** and **YNAME**, and **USERDEFINED** characteristics applied for **PXNAME_I** and **PYNAME_I**.

COULOMB-USERDEFINED-COULOMB : Scaling according to **COULOMB** for **XNAME**, **YNAME** and **PYNAME_I**, and **USERDEFINED** characteristics applied for **PXNAME_I**.

COULOMB-COULOMB-USERDEFINED : Scaling according to **COULOMB** for **XNAME**, **YNAME** and **PXNAME_I**, and **USERDEFINED** characteristics applied for **PYNAME_I**.

USERDEFINED-USERDEFINED-USERDEFINED : The curves as defined by the user are applied for all material curves in the local x- and y-directions. Equivalent to specifying **USERDEFINED** only.

USERDEFINED-COULOMB-USERDEFINED : Scaling according to **COULOMB** for **PXNAME_I**, and **USERDEFINED** characteristics applied for **XNAME**, **YNAME** and **PYNAME_I**.

USERDEFINED-USERDEFINED-COULOMB : Scaling according to **COULOMB** for **PYNAME_I**, and **USERDEFINED** characteristics applied for **XNAME**, **YNAME** and **PXNAME_I**.

USERDEFINED-COULOMB-COULOMB : Scaling according to **COULOMB** for **PXNAME_I** and **PYNAME_I**, and **USERDEFINED** characteristics applied for **XNAME** and **YNAME**.

NX: Number of penetration-dependent penetration-dependent material curves in local x-direction. Optional.

NY: Number of penetration-dependent penetration-dependent material curves in local y-direction. Optional.

PX_1: Penetration value at which the first penetration-dependent material curve in local x-direction is valid. Not specified for **NX=0**. Optional. (unit: L).

PXNAME_1: Local x-direction first penetration-dependent material curve name. The curve must be hyper-elastic. Not specified for **NX=0**. Optional.

PX_NX: Penetration value at which the last penetration-dependent material curve in local x-direction is valid. Not specified for **NX=0,1**. Optional. (unit: L).

PXNAME_NX: Local x-direction last penetration-dependent material curve name. The curve must be hyper-elastic. Not specified for **NX=0,1**. Optional.

PY_1: Penetration value at which the first penetration-dependent material curve in local y-direction is valid. Not specified for **NY=0**. Optional. (unit: L).

PYNAME_1: Local y-direction first penetration-dependent material curve name. The curve must be hyper-elastic. Not specified for **NY=0**. Optional.

PY_NY: Penetration value at which the last penetration-dependent material curve in local y-direction is valid. Not specified for **NY=0,1**. Optional. (unit: L).

PYNAME_NY: Local y-direction last penetration-dependent material curve name. The curve must be hyper-elastic. Not specified for **NY=0,1**. Optional.

For other element types than CONT126, the only optional parameter that will be applied is **XYCHOICE** with value equal to either **COULOMB** or **USERDEFINED** as shown in the below example.

EXAMPLE:

```
#        name     type    mux   muy  xname yname  zname  xychoice
MATERIAL bellm1   contact 0.5   1.0   matx  maty  matz1  coulomb
MATERIAL body2    contact 0.5   1.0   matx  maty  matz2
MATERIAL roller1  contact 0.3   0.3   matx  maty  matz3  userdefined
MATERIAL roller2  contact 0.3   0.3   matx  maty  matz4  userdefined
MATERIAL roller3  contact 0.3   0.3   matx  maty  matz5
```

For the CONT126 element type, the optional parameters must either consist of only **XYCHOICE** or all of the optional parameters as shown in the examples below. Regarding **XYCHOICE**, the first word governs the **XNAME** and **YNAME** curves, the second word governs the **PXNAME_I** curves and the third word governs the **PYNAME_I** curves. This is convenient when describing soil resistance where one part is related to **COULOMB** and the second part is related to earth pressure resistance better described by a **USERDEFINED** characteristic depending on the penetration. For such cases it is recommended to use two curves as illustrated in Fig. 2.30.

An arbitrary number of penetration-dependent curves **PXNAME_I** and **PYNAME_I** may be given and BFLEX2010 will interpolate the force and the stiffness based on the actual penetration. Zero force is always assumed at zero penetration. Extrapolation of the force and stiffness is done if the actual penetration exceeds the largest user-defined penetration value. Hence, very large force and stiffness magnitudes may result if the actual penetration exceeds the largest user-defined penetration value significantly. The sign of the user-defined penetration values are of no importance because BFLEX2010 applies the absolute value in the computations.

The actual penetration includes the contributions from the soil elastic penetration and the element initial displacement defined by the **INISTR** card in Section 2.25.

EXAMPLE:

```
#         name    type     mux   muy xname yname zname   xychoice
MATERIAL soil1   contact 0.5   1.0 soilx soily soilz   coulomb-userdefined-userdefined
#
#  Nx   Ny   px1   pxname1   py1   pyname1   py2   pyname2
    1    2   0.1   soilx_1   0.1   soily_1   0.2   soil_y2
```

EXAMPLE:

```
#         name    type     mux   muy   xname   yname   zname   xychoice
MATERIAL soil1   contact 0.5   1.0   soilx   soily   soilz   userdefined
```

If the optional parameter **XYCHOICE**=**USERDEFINED** is omitted, the material curves in the x- and y-directions are to be defined as a consecutive number of points defining displacement versus unit force per length or unit force. This is because BFLEX2010 will scale the unit force value by μF_z or $\mu F_z L$, depending on the element type. Here, μ is the friction factor, F_z is the z-direction contact force (unit: F), and L is the length of the pipe element in contact.

For **XYCHOICE**=**USERDEFINED**, the x- and y-direction curves are to be defined as

a consecutive number of points defining displacement versus force per unit length or force, i.e. no scaling with μF_z is applied.

For the standard case, the material curves **XNAME**, **YNAME** and **ZNAME** need to be defined using the **HYCURVE** or **EPCURVE** options. The latter option should be applied in modelling of friction, otherwise the friction force reversal will not be correctly modelled. The material curve units depend on the element type, see Table 2.2.

The contact point y-displacement u_y^{cp} contains contributions from both pure y-translation u_y and y-translation induced by x-rotation θ_x according to,

$$u_y^{cp} = u_y + r\theta_x \tag{2.11}$$

where r is the pipe external radius.

The local y-force due to pipe-soil interaction is calculated as,

$$F_y\left(u_y^{cp}\right) = \mu_y \bar{f}_y\left(u_y^{cp}\right) F_z L \qquad \text{if } \textbf{XYCHOICE=COULOMB} \tag{2.12}$$

$$F_y\left(u_y^{cp}\right) = f_y\left(u_y^{cp}\right) L \qquad \text{if } \textbf{XYCHOICE=USERDEFINED} \tag{2.13}$$

where μ_y=**MUY**, $\bar{f}_y$ is the dimensionless force per unit length defined by **YNAME**, F_z is the force in the local z-direction (unit: F), L is the length of the pipe element in contact, f_y is the force per unit length defined by **YNAME**, and u_y^{cp} is defined in Eq. (2.11). The local x-moment due to pipe-soil interaction is calculated as,

$$M_x = rF_y\left(u_y^{cp}\right) \tag{2.14}$$

which means that x-moment will be induced for rotation about the x-axis and y-displacement such that rolling contact interaction will be correctly handled.

Note that for the element type CONT126, the x-moment will be set to zero and the second term in Eq. (2.11) will be removed if the option **IGAP=2** is applied for the **CONTINT** card in Section 2.5.

The local x-direction for seabed contact elements is directed along the current x-axis of the element system, and will undergo the same rotations as the attached pipe node during the analysis. Hence, the element initial x-axis as defined by the **ELORIENT** card must be directed along the pipe element initial x-axis in order to get the local x- and y-force directed along and transverse to the pipeline, respectively, and to get the local x-moment acting about the pipeline x-axis. The local z-direction is adjusted to coincide with the radial direction of the pipe cross-section provided that the element initial x-axis and the pipe element initial x-axis coincide, and the local y-direction is computed by the cross product of local z-direction unit vector and the local x-direction unit vector.

For other contact element types, the local x-direction coincides with the current pipe axial direction, and the local y-direction is determined by the cross product of the contact normal direction unit vector and the local x-direction unit vector.

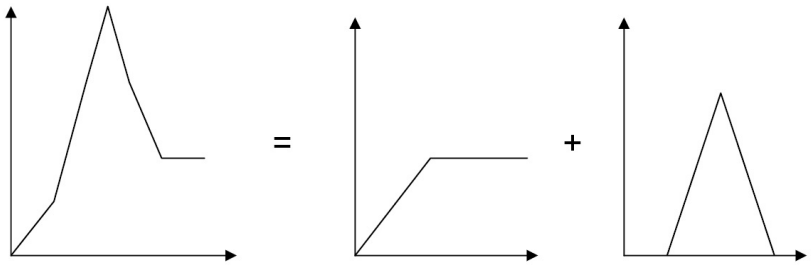


Figure 2.30: Material curve combination.

2.26.10 ISOCONTACT (CONT130, CONT152, CONT164)

MATERIAL MNAME ISOCONTACT MUXY XYNAME ZNAME

MUXY: Friction coefficient in local xy-direction

XYNAME: xy-plane material curve name

ZNAME: z-direction material curve name

EXAMPLE:

```
#        name      type        rmyx     xyname   zname
MATERIAL rollmat   isocontact  1.0      rollxy   rollz
#
#        name    type    ihard  eps  sig
MATERIAL rollxy epcurve   1     0.0  0.0
                                1.0  2.5e7
```

The **ZNAME** material curve can be defined by the **EPCURVE** or **HYCURVE** options, while only the **EPCURVE** option is allowed for the **XYNAME** material curve.

The **XYNAME** material is to be defined as a consecutive number of points defining displacement versus unit force per length. This is because BFLEX2010 will scale the unit force value with μN, where μ is the friction factor and N is the normal direction force.

2.26.11 ISOKXYCONTACT (CONT164)

MATERIAL MNAME ISOKXYCONTACT MUXY KSTICK ZNAME

MUXY: Friction coefficient in local xy-direction (unit: -).

KSTICK: Stick stiffness in local xy-direction (unit: FL^{-1}).

ZNAME: z-direction material curve name

EXAMPLE:

```
#        name      type        rmyx     kstick    zname
MATERIAL rollmat   isocontact  1.0      2.5e7     rollz
```

The **ZNAME** material curve can be defined by both **EPCURVE** and **HYCURVE**.

A standard Coulomb friction model is applied where the change of friction force in the stick state is equal to the product of the tangential displacement increment and **KSTICK**. As opposed to the **ISOCONTACT** option, the stick stiffness is constant and independent of the normal direction contact force. This may enhance the convergence properties in situations where large normal direction contact forces occur, such as for contact between hard materials or for large contact force spikes at initial contact.

2.26.12 FRICONTACT (HCONT453, 454, 463, 464 and 473)

MATERIAL MNAME FRICONTACT TYPE MUS MUD KEL1 KEL2 FRAC KDYN C1 C2 DIM KSURF

TYPE: Friction model type, 4=Static friction model describing a smooth transition from static to dynamic friction, 5 dynamic friction model with velocity dependent smooth transition to dynamic friction, 6=Coulomb model. Default value: 6

MUS: Static friction coefficient. Default value: 0.135 Note that during manual modelling, when combining FRICONTACT with HCONT464 to represent the interfacing contacts of a HSHEAR364 helix layer, see Section 2.20.13, a non-zero friction coefficient, MUS, will activate a friction moment model for the layer given by:

$$M_f = \sum_{i=1}^{2} f_{zi} \frac{\mu 2R^2 \cos\alpha}{\pi \tan\alpha} \tag{2.15}$$

where f_{zi} is the contact reaction per unit length along pipe axis at interface i, R is the helix radius, μ is the inputed static frition coefficient and α is the helix lay angle. The time at which friction is activated can be controlled by either of the

CONTINT or LAYERCONTACT inputs, see Section 2.5 and Section 2.20.11. For a flexible pipe (before introducing the viscoelastic part), this will result in a moment curvature relationship similar to **ITCODE** = 32 with reference to Section 2.10.1, however, making use of **KEL1** as the value of k in Eq. (2.6) when calculating the stick stiffness. Care should be taken with respect to not selecting a too low **KEL1** as noted below. For a flexible pipe, a large value of k will result in a moment curvature relationship similar to **ITCODE** = 32 with reference to Section 2.10.1. Eq. (2.15) is based on considering slip in one layer. For cases where multiple helices are separated by low strength plastic layers, the slip vector will be governed by the relative motions between the layers. This will cause a reduction of the available friction forces to build up stresses in each of the helix layers. This can be accounted for by imposing a reduction factor in Eq. (2.15), to be evaluated in each case.

MUD: Dynamic friction coefficient. Default value: 0.15

KEL1: Elastic shear stiffness in axial direction. Must be set with sufficient stiffness to reflect the plane surfaces remain plane condition. A recommended minmum value is $12\frac{EA\sin^2\alpha}{R^2}$ where EA is the axial stiffness of the associated helix element and R is the helix radius. Default: 1.010^9 Unit: $\frac{N}{m^2}$.

KEL2: Elastic shear stiffness in transverse direction. Default: 1.010^9 Unit:$\frac{N}{m^2}$

FRAC: The ratio between the transition slip distance (from static friction to dynamic friction) and the elastic distance (from zero to static friction) Default: 0.1

KDYN: Stiffness after the transition to dynamic friction. This stiffness might be set to a small positive value to improve numerical stability. Default: 0.0

C1: Anisotrophy factor in longitudinal direction Default: 1.0 (isotropic)

C2: Anisotrophy factor in transverse direction Default: 1.0 (isotropic)

DIM: Dimension of model (1, 11, 12, -1, -11 -12= 1D 2, -2 = 2D). The negative values only applies for the Coulomb friction model TYPE=6. By specifying a postive value, the 1D Coulomb friction algorithm is according to Tianjiao Dai's work (Dai et al., 2017), whereas Vegard Longva's model (Longva and Sævik, 2013) is applied if a negative value is given. The value 11/-11 is applied to enforce an extra contact pressure term related to deep grooves in the elastic support along the helix. Then the total contact force is taken to be $f_c = f_z + k_{el2} * u_y$ where f_z is the vertical contact force and u_y is the lateral transverse motion relative to the helix path. The value 12/-12 is applied to enforce an extra contact pressure term related to transverse elastic support along the helix. Then the total contact force is taken to be $f_c = \sqrt{{f_z}^2 + (k_{el2} * u_y)^2}$ where f_z is the vertical contact force and u_y is the lateral transverse motion relative to the helix path Default: 1

KSURF: Surface stiffness Default: 1.510^{11} Unit:$\frac{N}{m^2}$ Typically the surface stiffness needs to be calculated as $k_{surf} = E\frac{b}{t} \cdot c_p$ where E is the Young's modulus and t the

thickness of the softest material at the interface. Then b is the wire width and c_p is a penalty factor to avoid surface penetration. $c_p = 10$ normally suffice, however, sensitivity analysis is recommended.

EXAMPLE:

```
#MATERIAL MNAME    MATMODEL  TYPE  MUS MUD   KEL1 KEL2 frac kdyn C1   C2  dim ksurf
MATERIAL contmfri1 fricontact d   0.25 0.20  2e8  2e8  0.2   d  1.0  2.0  2  1.5e10
```

2.26.13 FRICVISC (HCONT463 and 464)

MATERIAL MNAME FRICVISC KSTICK KTRANS KSURF MU C1 C2 A1

The model combines Coulomb friction with viscoelastic behaviour in the longituinal helix direction. The Coulomb model is based on fixed stiffness values for the tangential and normal directions (fixed stick and normal stiffness), see(Longva and Sævik, 2013). Then the visoelastic model includes linear and non-linear terms. The longitudinal tangential force q_1 in the stick condition is then given by:

$$q_1 = k_1 u_i + c_1 \dot{u}_1 + c_2 |\dot{u}_1|^a \dot{u}_1 \tag{2.16}$$

where k_1 is the longitudinal shear stiffness, u_1 represents the longitudinal displacement, $\dot{u}_1$ represent the longitudinal slip velocity whereas c_1, c_2, a are viscoelastic constants. Then in the lateral transverse direction, the force q_2 is calculated as:

$$q_2 = k_2 u_2 \tag{2.17}$$

where k_2 is the transverse elastic stiffness. Then in the slip condition, the maximum Coulomb friction force is calculated by $q_{1c} = \mu\sqrt{q_2^2 + q_3^2}$

KSHEAR: Shear stiffness. Default: 1.010^9 Unit:$\frac{N}{m^2}$ In lack of other information, the stick stiffness should be set with sufficient stiffness to reflect the plane surfaces remain plane condition. A recommended minmum value is $12\frac{EA\sin^2\alpha}{R^2}$ where EA is the axial stiffness of the associated helix element and R is the helix radius.

KTRANS: Transverse elastic stiffness Default: 0.0 (unit:$\frac{N}{m^2}$)

KSURF: Surface stiffness Default: 1.510^{11} (unit:$\frac{N}{m^2}$) Typically the surface stiffness needs to be calculated as $k_{surf} = E\frac{b}{t} \cdot c_p$ where E is the Young's modulus and t the thickness of the softest material at the interface. Then b is the wire width and c_p is a penalty factor to avoid surface penetration. $c_p = 10$ normally suffice, however, sensitivity analysis is recommended.

MU: Friction coefficient. Default value: 0.135 Note that during manual modelling, when combining FRICVISC with HCONT464 to represent the interfacing contacts of a HSHEAR364 helix layer, see Section 2.20.13, application of Coulomb friction will activate a friction moment model according to Eq. (2.15). The time at which friction is activated can be controlled by either of the CONTINT or LAYERCONTACT inputs, see Section 2.5 and Section 2.20.11. For a flexible pipe (before introducing the viscoelastic part), this will result in a moment curvature relationship similar to **ITCODE** = 32 with reference to Section 2.10.1, however, making use of **KSHEAR** as the value of k in Eq. (2.6). Care should be taken with respect to not selecting a too low **KSTICK** as noted above. The friction moment model is based on considering slip in one layer. For cases where multiple helices are separated by low strength plastic layers, the slip vector will be governed by the relative motions between the layers. This will cause a reduction of the available friction forces to build up stresses in each of the helix layers. This can be accounted for by imposing a reduction factor to **MU** that needs to be evaluated in each case. By specifying a negative value, the Coulomb friction effect will be switched off and the bending moment contribution from the HSHEAR364 layer will then be calculated based on Eq. (2.5) and Eq. (2.6) without considering the friction limit. Similiarly, for the HCONT463 case, the first term of (2.16) will then always be valid.

C1: Linear damping coefficient Default: 0.0 (unit:$\frac{Ns}{m^2}$)

C2: Nonlinear damping coefficient Default: 0.0 (unit:$\frac{Ns^{1+a}}{m^2}$)

A1: Exponent of non-linear damping law coefficient Default: 1.0

EXAMPLE:

```
#MATERIAL MNAME    MATMODEL   kstick ktrans ksurf mu    c1   c2  a1
MATERIAL contmfri1 frivisc    2e8    2e8    2e8   0.2   1e6  d   d
```

2.26.14 R_CONTACT (CONT126)

MATERIAL MNAME R_CONTACT MUX MUY MUTX XNAME YNAME ZNAME TXNAME [[XYCHOICE] [NX NY PX_1 PXNAME_1 ... PX_NX PXNAME_NX PY_1 PYNAME_1 ... PY_NY PYNAME_NY]]

MUX: Friction coefficient in local x-direction (unit: -).

MUY: Friction coefficient in local y-direction. Negative value will provide isotropic friction based on **MUX** (unit: -).

MUTX: Friction coefficient in local y-direction due to x-rotation (unit: -).

XNAME: Local x-direction material curve name.

YNAME: Local y-direction material curve name.

ZNAME: Local z-direction material curve name.

TXNAME: Material curve defining local x-rotation versus the local y-direction force applied for calculating the x-moment.

XYCHOICE: Enables overruling of the default Coulomb friction model with userdefined characteristics for the local x-direction, the local y-direction and the local x-rotation. Optional. Allowable values:

COULOMB : The **XNAME** and **PXNAME_I** curves are scaled by the product of the local z-direction force and **MUX**, while the **YNAME** and **PYNAME_I** curves are scaled by the product of the local z-direction force and **MUY** The **TXNAME** curve is scaled by the product of the local z-direction force, the friction coefficient **MUTX** and the pipe external radius. This is the default behaviour.

USERDEFINED : The **XNAME**, **YNAME**, **TXNAME**, **PXNAME_I** and **PYNAME_I** curves as defined by the user are applied.

COULOMB-COULOMB-COULOMB : Scaling according to **COULOMB** is applied for all material curves for the local x- and y-directions and the local x-rotation. Equivalent to specifying **COULOMB** only.

COULOMB-USERDEFINED-USERDEFINED : Scaling according to **COULOMB** for **XNAME**, **YNAME** and **TXNAME**, and **USERDEFINED** characteristics applied for **PXNAME_I** and **PYNAME_I**.

COULOMB-USERDEFINED-COULOMB : Scaling according to **COULOMB** for **XNAME**, **YNAME**, **TXNAME** and **PYNAME_I**, and **USERDEFINED** characteristics applied for **PXNAME_I**.

COULOMB-COULOMB-USERDEFINED : Scaling according to **COULOMB** for **XNAME**, **YNAME**, **TXNAME** and **PXNAME_I**, and **USERDEFINED** characteristics applied for **PYNAME_I**.

USERDEFINED-USERDEFINED-USERDEFINED : The curves as defined by the user are applied for all material curves for the local x-direction, the local y-direction and the local x-rotation. Equivalent to specifying **USERDEFINED** only.

USERDEFINED-COULOMB-USERDEFINED : Scaling according to **COULOMB** for **PXNAME_I**, and **USERDEFINED** characteristics applied for **XNAME**, **YNAME**, **TXNAME** and **PYNAME_I**.

USERDEFINED-USERDEFINED-COULOMB : Scaling according to **COULOMB** for **PYNAME_I**, and **USERDEFINED** characteristics applied for **XNAME**, **YNAME**, **TXNAME** and **PXNAME_I**.

USERDEFINED-COULOMB-COULOMB : Scaling according to **COULOMB** for **PXNAME_I** and **PYNAME_I**, and **USERDEFINED** characteristics applied for **XNAME**, **YNAME** and **TXNAME**.

NX: Number of penetration-dependent material curves in local x-direction. Optional.

NY: Number of penetration-dependent material curves in local y-direction. Optional.

PX_1: Penetration value at which the first penetration-dependent material curve in local x-direction is valid. Positive for penetration into the seabed. Not specified for **NX=0**. Optional. (unit: L).

PXNAME_1: Local x-direction first penetration-dependent material curve name. The curve must be hyper-elastic. Not specified for **NX=0**. Optional.

PX_NX: Penetration value at which the last penetration-dependent material curve in local x-direction is valid. Positive for penetration into the seabed. Not specified for **NX=0,1**. Optional. (unit: L).

PXNAME_NX: Local x-direction last penetration-dependent material curve name. The curve must be hyper-elastic. Not specified for **NX=0,1**. Optional.

PY_1: Penetration value at which the first penetration-dependent material curve in local y-direction is valid. Positive for penetration into the seabed. Not specified for **NY=0**. Optional. (unit: L).

PYNAME_1: Local y-direction first penetration-dependent material curve name. The curve must be hyper-elastic. Not specified for **NY=0**. Optional.

PY_NY: Penetration value at which the last penetration-dependent material curve in local y-direction is valid. Positive for penetration into the seabed. Not specified for **NY=0,1**. Optional. (unit: L).

PYNAME_NY: Local y-direction last penetration-dependent material curve name. The curve must be hyper-elastic. Not specified for **NY=0,1**. Optional.

The purpose of **R_CONTACT** is to remove the x-moment dependency on the y-displacement. This is done by introducing the following modifications as compared to the **CONTACT** material type in Section 2.26.9:

- A separate curve **TXNAME** defines the y-direction force applied for calculating the x-moment.
- The local x-rotation is applied as the only displacement variable for the x-moment. The curve **TXNAME** is therefore defined in terms of the local x-rotation as the argument.
- The local x-rotation does not contribute to contact point displacement u_y^{cp} in the local y-direction. This means that the second term on the right-hand side of Eq. (2.11) is removed.

The above modications enable modelling of pipe-soil skin friction effects in situations with non-zero x-rotation and zero y-displacement. For instance, consider a pipe that is penetrated into the seabed and subjected to a x-rotation without displacing in the y-direction. Here, the skin friction effect will induce a x-moment and zero force in the local y-direction, with non-zero x-rotation and zero y-displacement. This effect is not possible to model with the **CONTACT** model in Section 2.26.9.

The **R_CONTACT** model gives zero x-moment for a case with zero x-rotation and non-zero y-displacement, however, the y-force will still be correctly modelled by the **YNAME** curve. The model shall therefore not be used for modelling of the x-moment for a pipe that displaces in the y-direction. In that case the **CONTACT** model in Section 2.26.9 shall be applied.

The local y-force due to pipe-soil interaction is calculated as,

$$F_y = \mu_y \bar{f}_y\left(u_y\right) F_z L \qquad \text{if } \textbf{XYCHOICE}=\textbf{COULOMB} \tag{2.18}$$

$$F_y = f_y\left(u_y\right) L \qquad \text{if } \textbf{XYCHOICE}=\textbf{USERDEFINED} \tag{2.19}$$

where μ_y=**MUY**, $\bar{f}_y$ is the dimensionless force per unit length defined by **YNAME**, F_z is the force in the local z-direction (unit: F), L is the length of the pipe element in contact, f_y is the force per unit length defined by **YNAME**, and u_y is the local y-displacement of the pipe node. Note that the y-displacement consists only of pure translation, i.e. the x-rotation coupling term in Eq. (2.11) is not included. The x-moment only depends on the x-rotation θ_x and is calculated as follows,

$$M_x = \mu_{tx} \bar{f}_{ty}\left(\theta_x\right) F_z r L \qquad \text{if } \textbf{XYCHOICE}=\textbf{COULOMB} \tag{2.20}$$

$$M_x = f_{ty}\left(\theta_x\right) r L \qquad \text{if } \textbf{XYCHOICE}=\textbf{USERDEFINED} \tag{2.21}$$

where μ_{tx}=**MUTX**, $\bar{f}_{ty}$ is the dimensionless y-force per unit length defined by **TXNAME**, F_z is the force in the local z-direction (unit: F), r is the pipe external radius, L is the length of the pipe element in contact, and f_{ty} is the y-force per unit length defined by **TXNAME**. Note that the x-moment will be set to zero if the option **IGAP**=**2** is applied for the **CONTINT** card in Section 2.5.

The optional parameters must either consist of only **XYCHOICE** or all of the optional parameters as shown in the examples below. Regarding **XYCHOICE**, the first word governs the **XNAME**, **YNAME** and **TXNAME** curves, the second word governs the **PXNAME_I** curves and the third word governs the **PYNAME_I** curves. This is convenient when describing soil resistance where one part is related to **COULOMB** and the second part is related to earth pressure resistance better described by a **USERDEFINED** characteristic depending on the penetration. For such cases it is recommended to use two curves as illustrated in Fig. 2.30.

An arbitrary number of penetration-dependent curves **PXNAME_I** and **PYNAME_I** may be given and BFLEX2010 will interpolate the force and the stiffness based on the actual penetration. Zero force is always assumed at zero penetration. Extrapolation of the force and stiffness is done if the actual penetration exceeds the largest user-defined penetration value. Hence, very large force and stiffness magnitudes may result if the actual penetration exceeds the largest user-defined penetration value significantly. The sign of the user-defined penetration values are of no importance because BFLEX2010 applies the absolute value in the computations.

The actual penetration includes the contributions from the soil elastic penetration and the element initial displacement defined by the **INISTR** card in Section 2.25.

EXAMPLE:

```
#        name    type       mux  muy mutx  xname yname zname txname
MATERIAL soil1   r_contact 0.5  1.0  0.7   soilx soily soilz soiltx
#
# xychoice                        Nx  Ny  px1  pxname1  py1  pyname1  py2  pyname2
coulomb-userdefined-userdefined   1   2   0.1  soilx_1  0.1  soily_1  0.2  soil_y2
```

EXAMPLE:

```
#        name      type       mux  muy   mutx xname  yname  zname   txname  xychoice
MATERIAL soil1     r_contact 0.5  1.0   0.5  soilx  soily  soilz1  soiltz
MATERIAL soil2     r_contact 0.5  1.0   0.5  soilx  soily  soilz2  soiltz  coulomb
MATERIAL roller1   r_contact 0.3  0.3   0.5  soilx  soily  soilz3  soiltz
MATERIAL roller2   r_contact 0.3  0.3   0.5     fx     fy  soilz4     mtz  userdefined
MATERIAL roller3   r_contact 0.3  0.3   0.5     fx     fy  soilz5     mtz  userdefined
```

If the optional parameter **XYCHOICE**=**USERDEFINED** is omitted, the material curves for the local x-direction and the local y-direction are to be defined as a consecutive number of points defining displacement versus unit force per length. This is because BFLEX2010 will scale the unit force by $\mu_x F_z L$ and $\mu_y F_z L$ for the local x- and y-directions, respectively. Likewise, the x-rotation curve is to be defined as a consecutive number of points defining x-rotation versus unit force per length. BFLEX2010 will then scale the unit force by $\mu_{tx} F_z L r$ to give the x-moment. Here, μ_x, μ_y and μ_{tx} are the friction factors for the local x-direction, y-direction and x-rotation, F_z is the local z-direction contact force (unit: F), r is the pipe external radius, and L is the length of the pipe element in contact.

For **XYCHOICE**=**USERDEFINED**, the x- and y-direction curves are to be defined as a consecutive number of points defining displacement versus force per unit length, i.e.

no scaling with μF_z is applied. Likewise, the x-rotation curve is to be defined as a consecutive number of points defining x-rotation versus a y-direction force per unit length that BFLEX2010 will multiply by the pipe external radius to give the x-moment.

specified for **XYCHOICE**.

For the standard case, the material curves **XNAME**, **YNAME**, **ZNAME** and **TXNAME** need to be defined using the **HYCURVE** or **EPCURVE** options. The latter option should be applied in modelling of friction, otherwise the friction force reversal will not be correctly modelled. The material curve units are defined in Table 2.2. In particular, it should be noted that the **TXNAME** material curve is to be defined as a consecutive number of points defining x-rotation (unit: radian) versus a y-direction force or unit force which BFLEX2010 multiplies by the pipe external radius to give the x-moment.

The local x-direction is directed along the current x-axis of the element system, and will undergo the same rotations as the attached pipe node during the analysis. Hence, the element initial x-axis as defined by the **ELORIENT** card must be directed along the pipe element initial x-axis in order to get the local x- and y-force directed along and transverse to the pipeline, respectively, and to get the local x-moment acting about the pipeline x-axis. The local z-direction is adjusted to coincide with the radial direction of the pipe cross-section provided that the element initial x-axis and the pipe element initial x-axis coincide, and the local y-direction is computed by the cross product of local z-direction unit vector and the local x-direction unit vector.

2.26.15 HYPERELASTIC (FLEXCROSS)

MATERIAL MNAME HYPERELASTIC POISS RHO0 TALFA TECOND

HEATC EPS SIGMA
... id...

POISS: Poisson's ratio (-)

RHO0: Density (dummy) (unit: ML^{-3})

TALFA: Temperature elongation coeff. (unit K^{-1})

TECOND: Thermal conductivity (dummy) (unit: $\mathrm{ET}^{-1}\mathrm{L}^{-1}\mathrm{TE}^{-1}$)

HEATC: Heat capacity (dummy) (unit: $\mathrm{E}^{-1}\mathrm{M}^{-1}\mathrm{TE}^{-1}$)

EPS: Strain

SIGMA: Stress (unit: FL^{-2}) values describing the material curve for both positive and negative values of strain and stress.

EXAMPLE:

```
#        name     type          poiss ro   talfa   tecond heatc eps   sigma
MATERIAL pipemat3 hyperelastic 0.3   7850 1.17e-5 50     800   0       0
                                                              0.005 450
                                                              1.0   700
#
```

2.26.16 GENSPRING (SPRING137)

MATERIAL MNAME GENSPRING XNAME YNAME ZNAME RXNAME RYNAME RZNAME [XYCHOICE]

XNAME: x-direction material curve name

YNAME: y-direction material curve name

ZNAME: z-direction material curve name

RXNAME: x-rotation material curve name

RYNAME: y-rotation material curve name

RZNAME: z-rotation material curve name

XYCHOICE: Friction force scaling is introduced by specifying **COULOMB**. For this option, the parameter **ICOULCNTR** in Section 2.20.7 must be set, which governs whether the friction force is activated in the xy-plane or only in the x-direction. By selecting **USERDEFINED**, the forces in the x- and y-directions are taken from the userdefined material curves without any scaling, which also is default behavior when the optional parameter **XYCHOICE** is omitted.

EXAMPLE:

```
#
#        name     type       spr1     spr2  spr3    spr4   spr5    spr6    xychoice
MATERIAL vessel1 genspring surgesp2 yawsp heavesp rollsp pitchsp swaysp  userdefined
```

It is noted that the material curves need to be defined using the **EPCURVE** or **HYCURVE** options in Sections 2.26.4 and 2.26.5, respectively. For both cases the curve is to be defined as a consecutive number of points defining displacement/rotation versus force/moment. As explained in Section 2.20.7, material curves that are scaled according to the **COULOMB** option should be defined by the **EPCURVE** option, otherwise reversal of the friction force will not be correctly modelled.

2.26.17 HELSPRING (HELSPR437)

MATERIAL MNAME HELSPRING AXFRIC XMATNAME XYCHOICE

AXFRIC: Axial coefficient of friction.

XMATNAME: Axial x-direction material curve name, using ep- or hycurve.

XYCHOICE: Enables the user to overrule the Coulomb friction concept by userdefined characteristics i x- and y - directions by introducing the parameter **USERDEFINED**. The default value is **COULOMB**.

2.27 MOVE_GROUP - move group during autostart

In order to enable complex structures to be moved by the autostart option, the **MOVE_GROUP** command enables the user to move the strucure equal to a given node as part of the autostart. The following format is applied.

MOVE_GROUP NOD1 ELGR [.. ELGR]

where

NOD1: Node ID. If a negative **NOD1** is given, then the nodes are incremented by 1 for each element node. The autostart logic is working such that all nodes having same coordinates as the autostart ones will be moved accordingly. The negative **NOD1** can be used if the user wants to force another node system being eccentric relative to the autostart node system to be moved accordingly.

ELGR: Element group name.

EXAMPLE:

```
#
# Contact interface data:
#-------------------------------------------------------------------------
#            NODEID   Group name 1    Group name 2
MOVE_GROUP   101      Tee-structure Templatestructure
```

2.28 NOCOOR - Nodal coordinates of model

By the **NOCOOR** command, the user can specify the initial coordinates of the elements. The nodal coordinates are defined by a card starting as follows:

```
NOCOOR TYPE ...
```

where

TYPE: Type of coordinate definition. Three values are valid:

COORDINATES: Nodal points are given in global coordinates system.

POLAR: Nodal points are given in a polar coordinates system for **HSHEAR352**. Local origo and orientation of the polar system must be specified. Nodal coordinates will be described along local x-axis by defining radius, angles and distance along the x-axis for each node.

ROTDISP: Enables the user to rotate and displace a given set of coordinates

If option **COORDINATES** is selected, the **NOCOOR** card has the following elements:

```
NOCOOR COORDINATES NODE XCOR YCOR ZCOR
                          [ ..    ..     ..     .. ]
[REPEAT N NODINC XINC YINC ZINC]
```

NODE: Node ID number of a node.

XCOR: x-coordinate.

YCOR: y-coordinate.

ZCOR: z-coordinate.

Under option **COORDINATES**, the **REPEAT** card has the following parameters:

N: Number of repeats (total number of times the nodal list defined above is repeated, including the original nodal list).

NODINC: Nodal increment.

XINC: Increment in x-coordinate.

YINC: Increment in y-coordinate.

ZINC: Increment in z-coordinate.

If option **POLAR** is selected, the **NOCOOR** card has the following elements:

```
NOCOOR POLAR X0 Y0 Z0 BETA1 BETA2 BETA3 R NODE XCOR THETA
                                              [ ..    ..     .. ]
[REPEAT N NODEINC XINC THETAINC]
```

X0: x-coordinate of local system.

Y0: y-coordinate of local system.

Z0: z-coordinate of local system.

BETA1: Rotation about x for local system.

BETA2: Rotation about y for local system.

BETA3: Rotation about z for local system.

R: Distance from local x-axis.

NODE: Node ID number of a node.

XCOR: x-coordinate along local x-axis.

THETA: Angle of node coordinate.

Under option **POLAR** the **REPEAT** sequence has the following parameters:

N: Number of repeats (total number of times the nodal list defined above is repeated, including the original nodal list).

NODINC: Nodal increment.

XINC: Increment along local x-axis.

THETAINC: Increment in angle.

If option **ROTDISP** is selected, the **NOCOOR** card has the following elements.

NOCOOR ROTDISP TX TY TZ XINC YINC ZINC

NODE: Node ID number of a node.

TX: Tait–Bryan angle around x axis.

TY: Tait–Bryan angle around the intrinsic y axis.

TZ: Tait–Bryan angle around the intrinsic z axis.

XINC: x-increment.

YINC: y-increment.

ZINC: z-increment.

An arbitrary sequence of nodes may be given, both for **POLAR** and **COORDINATES**, as indicated by the dots. However, the value of **NODE** must be constantly increasing. If **NODE** increases with more than 1, linear interpolation is used to create intermediate nodal coordinates.

By including the **REPEAT** command the **NOCOOR** sequence is repeated so that a total of **N** sequences is defined. The **REPEAT** command can be nested. Each instance will repeat all nodes which are defined so far in the **NOCOOR** card, including the nodes generated with **REPEAT**s above.

The **ROTDISP** causes all nodes from the previous **ROTDISP** card or from the start of the **NOCOOR** sequence until the present **ROTDISP** card to be rotated and displaced with the given quantity.

EXAMPLE:

```
#
#                    nodnr  x    y    z
NOCOOR COORDINATES    1     0    0    0
                     11     0    0   -10
                     21     0    0   -15
                     41     0    0   -20
```

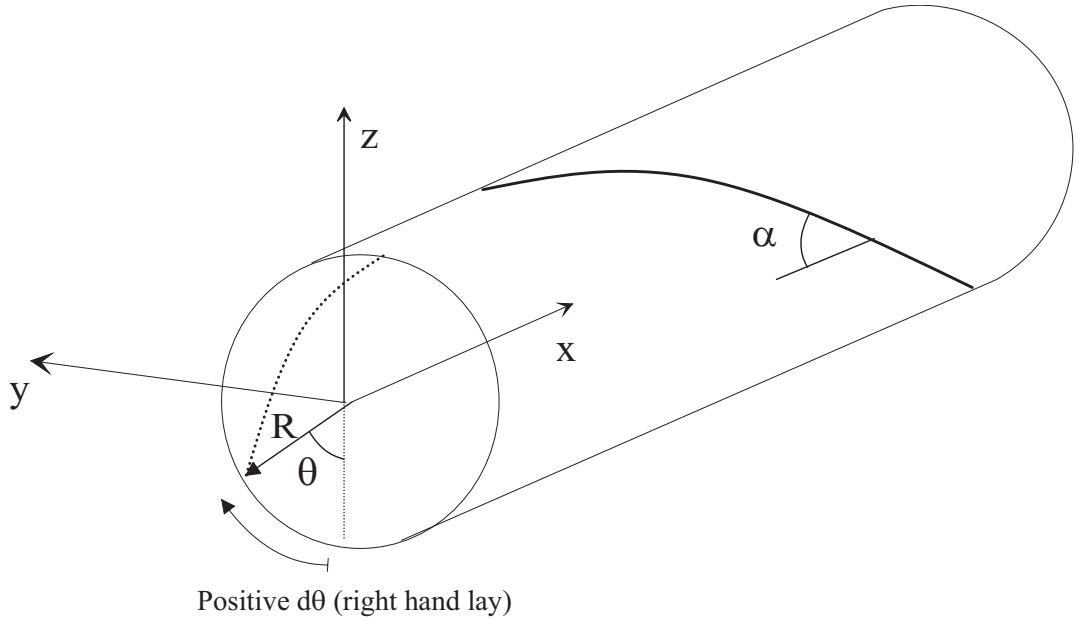


Figure 2.31: Polar coordinate definition.

2.29 NODPROP - node interpolated element properties

The **NODPROP** command allows the user to specify bellmouth or bend stiffener geometries, overruling the concept of constant geometry properties per group. The element properties in terms of radius and thickness is taken as the mean between the element nodes. The following format is applied:

```
NODPROP NODID RO RI
        ..    ..  ..
```

where

NODID: Node ID number j.
RO: Outer radius at node j (unit: L).

RI: Inner radius at node j (unit: L).

The **NODPROP** command allows adjusting the element geometry properties of PIPE31, PIPE33 and COMPIPE42 element types which are connected to the nodes at which **NODPROP** is specified.

The sequence of the three last numbers may be repeated for as many node numbers as defined. Values for missing node numbers will be determined by linear interpolation.

Note: The outer pipe diameter D_{op} of an element is calculated by averaging the radiuses from the **NODPROP** card according to $D_{op} = 0.5\,(2R_1 + 2R_2)$, where R_1 and R_2 are the radiuses at the element nodes. The outer diameter **ODP** specified in the **ELPROP** card will be over-ruled by D_{op}.

Note: The wrap diameter **ODW** specified in the **ELPROP** card in Sections 2.20.1, 2.20.2 is not adjusted. The hydrodynamic diameter is calculated as $D = (1-\eta)D_{op} + \eta D_{ow}$ where the wrap diameter D_{ow} is taken from **ODW** in the **ELPROP** input card and D_{op} is taken from the **NODPROP** card. Hence, **RKS**=**0.0** must be applied in the **ELPROP** card to get a hydrodynamic diameter corresponding to **RO** specified above.

Note: Regarding contact search, the inner diameter of the pipe is 2 **RI**. The outer diameter is adjusted in the following manner: The difference between the outer diameter and the structural diameter is calculated as Δ_p = **ODP** - 2 **RAD** + **TH** (refer to **ELPROP** card). Thereafter, the difference between the wrap diameter and the structural diameter is calculated as Δ_w = **ODW** - 2 **RAD** + **TH** (refer to **ELPROP** card). The maximum value is given by $\Delta_{\max} = \max(\Delta_p, \Delta_w)$. With this, the outer diameter at an element node applied in the contact search is set equal to 2 **RO** + $\Delta_{\max}$.

EXAMPLE:

```
#         nodid    Ro       Ri
NODPROP   1001     255      240
          1041     255      240
          1042     255      240
          1043     256      241
          1044     257      242
          1045     260      245
          1046     263      248
```

2.30 NOORIENT - orientation of nodes

The **NOORIENT** command is used to define the initial orientation of the nodes. As default these are oriented parallel to the global coordinate frame, i.e. if no **NOORIENT** card is present this will be the case. However, if constraint equations is to be introduced in a local coordinate system, the user is allowed to change the orientation to e.g. parallel to the element coordinate system, see Section 2.19. The orientation of the node may be defined by specifying a set of three consecutive Tait–Bryan angles (θ_x, θ_y, θ_z) that rotates the default coordinate system relative to the global coordinate system in the defined sequence, see Fig. 2.32. The format is similar to the **NOCOOR** command:

```
NOORIENT TYPE NODE X Y Z
                      [..      .. .. ..
[REPEAT N INC XINC YINC ZINC]
```

TYPE: Type of orientation. This parameter can have the values **COORDINATES** og **EULERANGLE**.

In case of **COORDINATES** the remaining parameters have the following meaning:

NODE: id of node.

X: x-coordinate of second normal vector (First normal vector is directed along the element axis.)

Y: y-coordinate of second normal vector.

Z: z-coordinate of second normal vector.

whereas if **TYPE** = **EULERANGLE**:

NODE: id of node.

X: Tait–Bryan angle around x-axis.

Y: Tait–Bryan angle around intrinsic y-axis.

Z: Tait–Bryan angle around intrinsic z-axis.

In both cases an arbitrary sequence may be given, as long as **NODE** is increasing. If the difference between two consequtive nodes is > 1, then linear interpolation is used to create intermediate nodal coordinates.

By including the **REPEAT** command the **NOORIENT** sequence is repeated **N** times:

N: Number of repetitions.

INC: Node increment.

XINC: x-coordinate/Tait–Bryan angle increment.

YINC: y-coordinate/Tait–Bryan angle increment.

ZINC: z-coordinate/Tait–Bryan angle increment.

EXAMPLE:

```
#
#          type             nodnr       x      y       z
NOORIENT   COORDINATES          1       0      1       0
                               11       0      1     -10
                               21       0      1     -15
                               41       0      1     -20
#
#         type              node       tx     ty      tz
NOORIENT EULERANGLE         3001       0      0       0
```

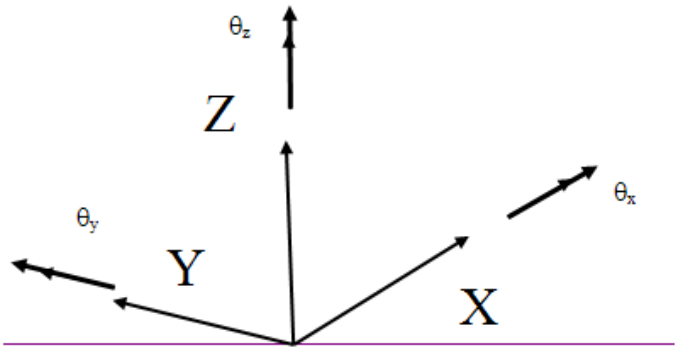


Figure 2.32: **NOORIENT** command.

2.31 PELOAD - external pressure and gravity loading

The external pressure and gravity loads are specified by the following format:

PELOAD PRESHIST GRAVHIST

where

PRESHIST: External pressure and buoyancy mass history number.
GRAVHIST: Dry mass history number.

Refer to the concepts of dry mass and dry buoyancy mass concepts applied in the **ELPROP**, see Section 2.20. This means at a given time t:

$$f_z(t) = m_{d0}\, T_d(t)\, g + (m_{d-b0} - m_{d0})\, T_b(t)\, g$$
$$p_e = \rho\, g\, (z_s - z)\, T_b(t)$$
$$m_d = m_{d0}\, T_d(t)$$

where:

f_z = gravity load (unit: FL^{-1})
m_{d0} = dry mass reference value as defined in **ELPROP** card (unit: ML-1)
$T_d(t)$ = dry mass history factor at time t,
as defined in the **THIST** refered to by **GRAVHIST**
m_{d-b0} = dry mass - buoyancy mass reference values as defined in **ELPROP** card
= submerged weight/gravity (unit: ML^{-1})
$T_b(t)$ = buoyancy mass history factor at time t,
as defined in the **THIST** refered to by **PRESHIST**
$m_d(t)$ = dry mass at a given time t
g = gravity constant (unit: LT^{-2})
ρ = seawater density (unit: ML^{-3})
p_e = external pressure (unit: FL^{-2})

For a typical installation analysis, the masses normally refers to the empty pipe condition, hence only one history is required to handle buoyancy and dry masses. However, if a free-span analysis is to be carried out for empty, waterfilled and operation conditions, the dry mass change whereas the buoyancy is constant. In that case two separate histories will be needed (the dry mass change whereas the buoyancy is constant).

EXAMPLE:

```
#
#             (gravity loading)
#       externpreshistnr     gravloadhistnr
PELOAD  100                  100
```

2.32 PILOAD - internal pressure load

The internal pressure is specified by the following format:

PILOAD HIST ELNR1 P1 [ELNR2 P2]

where

HIST: Load history number.
ELNR1: First element ID number.
P1: Internal pressure of first element (unit: FL^{-2}).
ELNR2: Last element ID number.

P2: Internal pressure of last element (unit: FL^{-2}). Linear interpolation is applied between intermediate elements.

If negative element numbers are given for the element types PIPE52, HSHEAR363 and HSHEAR364, the specified pressure will be handled as an external pressure. It is to be noted that this is the only way of specifying external pressure for these element types as the logic related to applying the sea element to provide external pressure from seawater has not been implemented. For element type PIPE52, two input cards, one describing internal pressure and one describing the external pressure is suffiecient and the pressures will be acting according to the flexible pipe cross-section input, see Section 2.10.1.

For the HSHEAR363 and HSHEAR364 element types the pressures must be specified for all element groups that are exposed to pressure and according to the following rules.

Note that for thinwall tubular elements, such as HSHEAR363, both internal and external pressure will act on the tubular mean radius. For thickwall version HSHEAR364 the internal pressure is applied in the inner radius, and external pressure on the outer radius.

The user can specify both internal and external pressure. This also applies for helices such that umbilical tubes can be handled by specifying internal pressure for the helix element group.

The following statements applies if the model represents a flexible pipe made by applying 353FLEXCROSS, see Section 2.10.2:

1. For all layers (element groups) $\geqslant$ ILAEXT, the same external pressure must be specified. Extenal pressure will then be applied to the outer surface of ILAEXT, and to both inside and outside of the layers outside of ILAEXT. Note that for thinwall elements, the mean radius is used so that inner and outer surface will be at the same radius.

2. For all layers (element groups) $\leqslant$ ILAINT, the same internal pressure must be specified. Internal pressure will then be applied to the inner surface of ILAINT, and to both inside and outside of the layers inside of ILAINT. Note that for thinwall elements, the mean radius is applied so that inner and outer surface will be at the same radius.

For manual input made to represent arbitrary helical structures, the following input rules applies:

1. For all layers (element groups) surrounded by an external pressure, both external and internal pressure of same magnitude must be specified

2. For the innermost layer exposed to external pressure, specify only external pres-

sure

3. For all layers (element groups) surrounded by an internal pressure, both external and internal pressure of same magnitude must be specified

4. For the outermost layer exposed to internal pressure, specify only internal pressure

EXAMPLE:

```
#      (internal pressure load)
#        loadhistnr  elnr1   p1     elnr2  p2
PILOAD   400         16001   69.0   16280  69.0
PILOAD   400         17001   69.0   17280  69.0
```

EXAMPLE:

```
#      (enternal pressure load)
#
#        loadhistnr  elnr1   p1     elnr2  p2
PILOAD   400         -16001   69.0   -16280  69.0
PILOAD   400         -17001   69.0   -17280  69.0
```

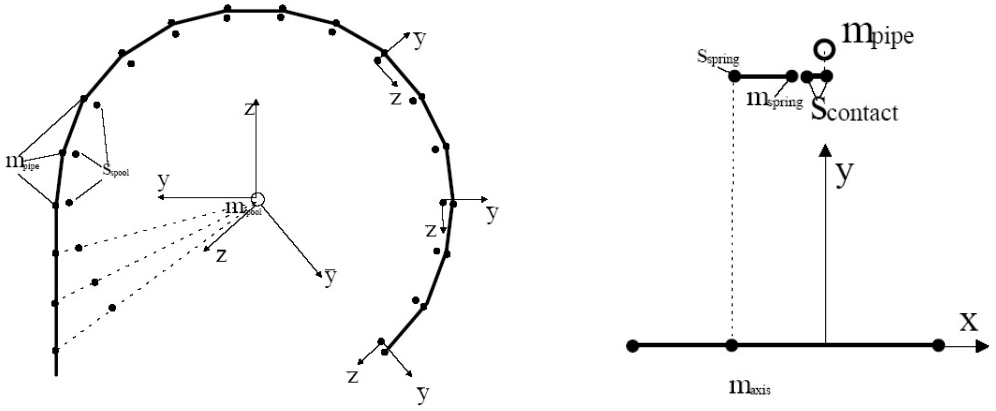
2.33 REEL - reeling and straightening simulation scenario definition

The **REEL** command is used to define the components needed in a reeling/straightening simulation. In addition to the pipe intended for reeling such an analysis must contain a spool and possible rolls to model a straightener.

Master-Slave technique is utilized in the modelling of reeling and hence constraint equations between the different components must be defined in addition to the **REEL**-command. This is done by the **CONSTR CONEQ**-command, see Sections 2.4 and 2.4.2.

The principle of the constraint formulation is illustrated in (a) in Fig. 2.33. The pipe group intended for reeling contains only master nodes, except for the first node, which is a slave of the spool axis. Each of the master nodes in the pipe has a corresponding slave node in the spool. This slave follows the rotation of the master in the spool and is always located "under" it's master in the pipe. The local coordinate system is always normal to the spool as indicated in the figure. When contact between a pipe node and the spool occurs the displacement normal to the spool is locked to the slave node. In order to monitor the contact force and have a criterion for disconnecting, the slave in the spool is connected to another slave by a stiff element. The second slave node of this "contact element" is locked to a spring (ordinary beam-element), as indicated in (b) in

Fig. 2.33. The contact force for each point can then be found as the shear force (local y-direction) of the beam. For a given negative contact force limit (or zero) the master node in the pipe disconnects from the slave node and the slave node connects to the master of the spring. The location of the slave node underneath the master node in the pipe is governed by a constraint equation between the slave node of the "spring" and the master node of the spool axis.



(a) View from side. (b) View from top.

Figure 2.33: Reeling.

The principle for the rolls is similar, but in this case only one master node in the pipe connects to one slave node in the roll. Hence only one "slave-element" needs to be defined, but all nodes in the pipe must be masters to this slave. The spring element may be the roll itself.

It should be noted that all constraint equations related to contact between pipe and spool/rolls are applied in the local coordinate system of the slave.

The **REEL** command has the following format:

REEL REELGRP ROPIPE SPOOLSPRGRP SPOOLCOGRP RSPOOL FCLIMSPOOL [ROLLGRP ROLLCOGRP RROLL FCLIMROLL]

where:

REELGRP: Name of the pipe group intended for REELING.

ROPIPE: Outer radius of the reeled pipe.

SPOOLSPRGRP: Name of group containing "spring elements" in spool.

SPOOLCOGRP: Name of group containing "contact elements" in spool.

RSPOOL: Radius of spool.

FCLIMSPOOL: Contact-force cut-off limit for spool.

ROLLGRP: Name of group for roll.

ROLLCOGRP: Name of group for "contact element" in roll.

RROLL: Radius of roll.

FCLIMROLL: Contact-force cut-off limit for spool.

The sequence **ROLLGRP ROLLCOGRP RROLL FCLIMROLL** can be repeated to model multiple rolls.

EXAMPLE:

```
#
#   Reeling pipe:
# Pipe ID     Radius
REEL    centerpipe  0.25
#
#       Spool:
#       Spr ID    Cont ID   Radius  fclim
        spring1   slave1    7.5     0.0
#
# Strightner:
#       Spr ID    Cont ID   Radius  fclim
        roll1     droll1    0.5     0.0
        roll2     droll2    0.5     0.0
        roll3     droll3    0.5     0.0
        roll4     droll4    0.5     0.0
```

2.34 TABLE - Table data

The table command is used for defining data arrays. The array must be rectangular and must contain at least two rows and two columns. Tables are for instance used for initial soil embedment in Section 2.20.15.

The format is as follows:

```
TABLE NAME NCOL
        T11 T12 .. T1_NCOL
        ..
        TN1 TN2 .. TN_NCOL
```

where

NAME: Name of the table.

NCOL: Number of columns in table. Must be larger than or equal to 2.

T11: Value of entry (1,1), first arguement value

T12: Value of entry (1,2), first function value of column 2.

T1_NCOL: Value of entry (1,ncol), first function value of last column.

TIJ: Value of entry (i,j).

TN1: Value of entry (n,1), last argument value.

TN2: Value of entry (n,2), last function value of column 2.

TN_NCOL: Value of entry (n,ncol), last function value of last column.

The first column defines the argument values, and the other columns define the function values. Linear interpolation of the function values are applied in between the argument values. If the argument is less than **T11**, the first function value of the column(s) will be applied. If the argument is larger than **TN1**, the last function value of the column(s) will be applied.

EXAMPLE:

```
#-------------------------------------------------------
#      name       ncol
#-------------------------------------------------------
TABLE  masscoef   2
#
# gap  cm
#
0.0    3.28
0.4    2.0
20.0   2.0
#-------------------------------------------------------
#
#-------------------------------------------------------
#      name       ncol
#-------------------------------------------------------
TABLE  loadcoef   4
#
# arg  f1    f2    f3
#
0.0    3.2   1.2   0.7
1.0    2.0   1.0   0.6
20.0   2.0   1.0   0.5
#-------------------------------------------------------
```

2.35 THIST - time histories

By the **THIST** command the user is allowed to describe the load histories related to the different loads specified. The following format is applied:

```
THIST NO T1 FAC1
          T2 FAC2
          .. ..
```

where

NO: History number.

T1: Time t1.

FAC1: Load factor for time t1.

T2: Time t2.

FAC2: Load factor for time t2

An arbitrary sequence may be given, and linear interpolation is applied between the time steps defined by the **TIMECO** card above. For $t \in$ [T1 , T2] the load factor will then be:

$$F = \frac{\text{FAC2}\,(t - \text{T1}) + \text{FAC1}\,(\text{T2} - t)}{\text{T2} - \text{T1}} \tag{2.22}$$

The load factor is multiplied with the load level given in the relevant load definition.

2.35.1 THIST_F - history from file

A special version of **THIST** can be applied to obtain user defined load histories from file. The following format applies:

```
THIST_F NO TFILE
```

where

NO: History number.

TFILE: file name containing the parameters t1,fac1,t2,fac2...,defined above

2.35.2 THIST_H - harmonic load

A special version of **THIST** can be applied to obtain harmonic loads. The following format applies:

```
THIST_H NO OMEGA PHASE
```

where

NO: History number.
OMEGA: Frequency, ω.
PHASE: Phase displacement, φ.

The load factor at any time t, is then defined by:

$$F = sin(\omega t + \varphi) \tag{2.23}$$

2.35.3 THIST_R - ramping loads smoothly on or off

A special version of **THIST** can be applied if loads should be ramped on or off in a smooth manner. This is in particular recomended for wave loads. The following format applies:

```
THIST_R NO START STOP RAMPTYPE FAC
          ..    ..    ..       ..
```

NO: History number.
START: Start time for ramping.
STOP: Stop time for ramping.
RAMPTYPE: Type of ramping, only **RAMPCOS** allowed.
FAC: Load factor after the ramping.

The shape of the ramping function is a half cosine period, starting at zero slope. Between ramping periods, the load factor is keept constant at the value obtained in the last ramping period. Before first ramping, the load factor is zero.

EXAMPLE:

```
#            weight and external pressure
#
#       no     time    fac empty pipe = 0.421
THIST   100    0       0.4210
               1       0.4210
               1.05    0.4210
               2       1.0000
               3       1.0000
               4       0.4210
               5       0.5376
              32.5     0.5376
```

```
#
#           temperature
#       no      time    fac
THIST   200     0       0.0
                5       0.0
               25      131.9
               32.5    131.9
#
#          no    filename
THIST_F   100    file100
#:
#
#       no      omega   phase
THIST_H 100     0.628   0.0
#
#
         no     start  stop  Ramptype   fac
THIST_R 100    10      20    RAMPCOS    1.0
               30      40    RAMPCOS    0.0
```

2.36 TIMECO - time control data

By the **TIMECO** command the user defines the analysis as a function of time. The format is as follows:

TIMECO T DT DTVI DT0 TYPE [STEPTYPE ITERCO ITCRIT MAXIT MAXDIV CONR]

where:

T: Time to simulate to.

DT: Time increment to be used to reach the required time.

DTVI: Time increment between each restart/visual storage to the .raf file.

DT0: Time increment between each zero setting of the accumulated convergence control vectors.

TYPE: The analysis type which may be **STATIC** or **DYNAMIC** .

STEPTYPE: Type of step control, optional parameter If **STEPTYPE** is given, **ITERCO**, **ITCRIT**, **MAXIT**, **MAXDIV** and **CONR** must also be given. **STEPTYPE** can have two different values:

MANUAL : Time stepping is carried out as normal, but given **MAXIT** overrules the one given in the **CONTROL** card.

AUTO : If required tolerance is not met within **MAXIT**, the step is subdivided in two parts. This is repeated until required tolerance is met within **MAXIT** or the maximum number of allowed sub-divisions **MAXDIV** is reached. BFLEX2010 will proceed with the new, smaller time step until next ordinary time step is reached. If, during the sub-stepping, the tolerance is not met the time step will be split and iteration start over from the last ordinary time step.

ITERCO: Iteration control parameter. Must be given if **STEPTYPE** is given.

NONE : usual

GO-ON : the program proceeds after the maximum number of iterations is reached

ITCRIT: Iteration criterion parameter, enables the user to select between alternative incremental norms scaled with respect to the associated accumulated norms. Must be given if **STEPTYPE** is given.

DISP : displacement norm is used

FORC : force norm is used

ENER : energy norm is used

ALL : all norms are used

MAXIT: Maximum number of iterations. Must be given if **STEPTYPE** is given.

MAXDIV: Maximum number of sub-divisions. Must be given if **STEPTYPE** is given.

CONR: Convergence radius. Recommended value is 10^{-5}–10^{-7}. Overrules the global convergence radius from the **CONTROL** card in Section 2.6. Must be given if **STEPTYPE** is given.

EXAMPLE:

```
#
#------------------------------------------------------------
# Analysis time control:
#------------------------------------------------------------
#        t     dt    dtvi  dt0    type
TIMECO   1     1     1.0 201.0  STATIC
#
#       Waterfilling
#
#        t     dt    dtvi    dt0     type  steptype iterco itcrit maxit maxdiv  conr
TIMECO   50.0  0.1     1.   201.0  DYNAMIC      auto go-on   disp     15  4     0.001
```

2.37 TLOAD - temperature loading

The temperature loading is specified by the following format:

TLOAD HIST ELNR1 T1 [ELNR2 T2]

where

HIST: Load history number.

ELNR1: First element ID number.

T1: Temperature of first element (unit: TE).

ELNR2: Last element ID number.

T2: Temperature of last element (unit: TE). Linear interpolation is applied for the intermediate elements.

EXAMPLE:

```
#            (temperature loading)
#       loadhistnr  elnr1  t1     elnr2  t2
TLOAD   300         16001  100.0  16280  100.0
```

2.38 VISRES - visual results

By including the **VISRES** card, visual presentation in XPOST is enabled. Only one **VISRES** card can be given, and all results can be listed in that card. If no **VISRES** card is present, only restart info will be stored on the .raf file. However, BFLEX2010POST can still be applied.

Most structural elements are visualized as tubulars, whereas most contact elements are visualized as lines.

However, CONT126 additionally includes shell elements to visualize the sea bottom.

The HSHEAR352 element is visualized by a rectangular cross-section along the helix whereas for HSHEAR353 both rectangular and tubular cross-sections are allowed for.

The format is as follows:

VISRES MODE FACTOR RESULT [...]

where

MODE: Result presentation mode:

INTEGRATION : FEM results according to the list defined below is enabled including consistent representation of all numerical elements of the FEM model in XPOST including the applied element and node numbers. In addition to

the list below finite element resultant force results of all pipe elements is allowed for. Other options are deprecated, but this value are kept for backwards compatibility.

FACTOR: Scaling factor for all radial quantities (scale pipe radius).

RESULT: Result types that will be stored, can be a list. Options are:

SIGMA-XX : σ_{xx} Longitudinal stress (cable elements) at mean surface (pipe elements).

SIGMA-YY : σ_{yy} Hoop stress at mean surface- thinwalled theory (pipe elements).

SIGMA-XY : σ_{xy} Shear stress at mean surface (pipe elements).

STRAIN-XX : ε_{xx} Longitudinal strain at mean surface - (pipe elements).

STRAIN-YY : ε_{yy} Hoop strain at mean surface (pipe elements).

STRAIN-XY : ε_{xy} Shear strain at mean surface (pipe elements).

ACCSTRAIN : ε_{tot} Total longitudinal strain (elastoplastic pipe elements).

SIGMA-XXI : σ_{xxi} Longitudinal stress at inner surface (elastic pipe elements).

SIGMA-YYI : σ_{yyi} Hoop stress at inner surface-thickwalled theory (elastic pipe elements).

SIGMA-XYI : σ_{xyi} Shear stress at inner surface (elastic pipe elements).

SIGMA-XXO : σ_{xxo} Longitudinal stress at outer surface (elastic pipe elements).

SIGMA-YYO : σ_{yyo} Hoop stress at outer surface-thickwalled theory (elastic pipe elements).

SIGMA-XYO : σ_{xyo} Shear stress at outer surface (elastic pipe elements).

VCONDIS-X : Contact element local x-displacement.

VCONDIS-Y : Contact element local y-displacement.

VCONDIS-Z : Contact element local z-displacement.

VCONFOR-X : Contact element local x-force (unit: See Table 2.1)

VCONFOR-Y : Contact element local y-force (unit: See Table 2.1)

VCONFOR-Z : Contact element local z-force (unit: See Table 2.1)

SIGMA-XX-C1 : Longitudinal stress in corner 1 of tensile armour. Only for PIPE52 elements.

SIGMA-XX-C2 : Longitudinal stress in corner 2 of tensile armour. Only for PIPE52 elements.

SIGMA-XX-C3 : Longitudinal stress in corner 3 of tensile armour. Only for PIPE52 elements.

SIGMA-XX-C4 : Longitudinal stress in corner 4 of tensile armour. Only for PIPE52 elements.

SIGMA-XX-AX : Axial stress in tendon. Only for PIPE52, uchshear352 and HSHEAR353 elements.

SIGMA-XX-MY : Longitudinal stress due to bending about weak axis. Only for PIPE52, uchshear352 and HSHEAR353 elements.

SIGMA-XX-MZ : Longitudinal stress due to bending about strong axis (i.e. about surface normal). Only for PIPE52, uchshear352 and HSHEAR353 elements.

For helix elements HELIX231, HELIX233, HELIX234 and HELIX235, **VCONDIS-X**, **VCONDIS-Z**, **VCONFOR-X** and **VCONFOR-Z** can be used to obtain the bending induced effects. These are defined by the contact material in the **ELPROP** card of the helix.

EXAMPLE:

```
#         type          scaling factor     result list
VISRES    integration   1                  sigma-xx sigma-yy
```

3 BFLEX2010POST Report Generator

3.1 General

BFLEX2010POST includes the following files:

prefix.2bpi the BFLEX2010POST Input File.

prefix.2bpl the BFLEX2010POST Log File where warnings and error messages are written.

References are given in the input file to the relevant .raf file and the output .mpf files. All output is given on MATRIXPLOT (.mpf) format, which are ASCII files that can be further processed by e.g. Matlab or Python scripts.

Note that results are available only at time steps which are stored on the .raf file.

When running BFLEX2010POST from the command line, the basis is to write the name referring to BFLEX2010POST in the prompt. In addition, the following optional command line arguments can be applied:

- Adding -n and the input file name prefix makes BFLEX2010POST execute the post-processing without further requesting the user to specify the file name.
- Adding -m and a float number representing the multiplication factor for scaling of the default local work memory size used by BFLEX2010POST.
- Adding -d and a float number representing the multiplication factor for scaling of the default memory size for retrieving the RAF file results from BFLEX2010.

The optional arguments above may be specified in an arbitrary order. The option -d may be required when BFLEX2010 also has been executed with the -m option as described in Section 1.7.

EXAMPLE:

```
command line: BFLEX2010POST  -n prefixname  -m 2.1  -d 4.4
```

3.2 Input data

All input is described on ASCII file format. The maximum line length is 136 characters and the maximum number of letters in one single text string is 32.

Comment text strings are defined by introducing # at the start of the line.

The following identifiers define the different data groups:

NOPLOT for NOdal history PLOTs

IPPLOT for Integration point history PLOTs

ELPLOT for ELement history PLOTs

GNPLOT for Global Nodal PLOTs

GLPLOT for Global Element PLOTs

RSPLOT for ReStart PLOTs

NRPLOT for Nodal Reaction PLOTs

FATIGUE for FATIGUE calculations

ENPLOT for ENvelope PLOTs

BFPOST for BFlex POSTprocessing

GRPLOT Element reaction plots

In the following the input data will be explained.

3.3 NOPLOT - NOdal history PLOTs

NOPLOT is linked to the nodal quantities in terms of accumulated nodal displacement and rotation only.

The **NOPLOT** format is:

NOPLOT RAFPRE MPFPRE XLEG XRES YLEG YRES FNODID LNODID XSCL YSCL

RAFPRE: The BFLEX2010.raf file name prefix. If the name is defined by a combination of lower case and upper case letters, use e.g. 'input-filename'

MPFPRE: The output .mpf file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'mpf-filename'

XLEG: The legend name for the x-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g 'Displacement (m)'

XRES: x-axis result type. The following result types are available:

LOADSTEP : Load step

TIME : Time

HISTN : History number with reference to BFLEX2010 input file, e.g HIST100

YLEG: The legend name for the y-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g. 'Displacement (m)'

YRES: y-axis result type. The result types available are according to the user selected results, see below:

NODISP-X: x-displacement (unit: L)

NODISP-Y: y-displacement (unit: L)

NODISP-Z: z-displacement (unit: L)

NOROT-X: x-displacement (unit: L)

NOROT-Y: y-displacement (unit: L)

NOROT-Z: z-displacement (unit: L)

FNODEID: First node ID number in visual model

LNODEID: Last node ID number in visual model

XSCL: Scaling factor to be used for x-axis (to convert to convenient unit)

YSCL: Scaling factor to be used for y-axis (to convert to convenient unit)

EXAMPLE:

```
#       Raf file name          plotfilename
NOPLOT  "bellmouthdpwumb11"  "belldpwumb11-sxx"
#
# x-legend     x-res    y-legend      y-res    fnodid lnodid xscl yscl
 "Angle (deg)" HIST200 "Stress (MPa)" NOROT-X 16550   16611  10   1
```

3.4 IPPLOT - Integration Point history PLOTs

IPPLOT is linked to the nodal selection defined in Section 2.38

The **IPPLOT** format is:

IPPLOT RAFPRE MPFPRE XLEG XRES YLEG YRES FELID LELID LSECID CSECID XSCL YSCL

RAFPRE: The BFLEX2010.raf file name prefix. If the name is defined by a combination of lower case and upper case letters, use e.g. 'input-filename'.

MPFPRE: The output .mpf file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'mpf-filename'.

XLEG: The legend name for the x-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g 'Displacement (m)'.

XRES: x-axis result type. The following result types are available:

LOADSTEP : Load step

TIME : Time

HISTN : History number with reference to BFLEX2010 input file, e.g HIST100

YLEG: The legend name for the y-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g. 'Displacement (m)'.

YRES: y-axis result type. The result types available are according to the user selected results, see Section 2.38.

FELID: First element ID number in visual model.

LELID: Last element ID number in visual model.

LSECID: Integration section number along element. Max 2 for PIPE31, PIPE34 and PIPE37. Max 3 for PIPE33, PIPE36, PIPE39, COMPIPE42 and CABLE111.

CSECID: Integration section number in cross-section. The total number is given in the control card of the SIMLA analysis.

XSCL: Scaling factor to be used for x-axis (to convert to convenient unit).

YSCL: Scaling factor to be used for y-axis (to convert to convenient unit).

EXAMPLE:

```
#       Raf file name plotfilename
IPPLOT  "bellmouth11" "bell11-sxx"
#
#x-legend     x-res   y-legend     y-res    fnodid lnodid felid lelid lsecid csecid xscl yscl
"Angle(deg)" HIST200 "Stress(MPa)" SIGMA-XX 16550 16611   1     5     10     1      1.0  1.0
```

3.5 ELPLOT - ELement history PLOTs

The **ELPLOT** format is:

ELPLOT RAFPRE MPFPRE XLEG XRES YLEG YRES FELID LELID XSCL YSCL [ELEND]

RAFPRE: The BFLEX2010.raf file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'input-filename'.

MPFPRE: The output .mpf file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'mpf-filename'.

XLEG: The legend name for the x-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g. 'Displacement (m)'.

XRES: x-axis result type. The following result types are available:

LOADSTEP : Load step

TIME : Time

HISTN : History number with reference to BFLEX2010 input file, e.g. HIST100

YLEG: The legend name for the y-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g. 'Displacement (m)'.

YRES: y-axis result type. The following result types are available:

ELFORCE-X : Fx force (unit: F)

ELFORCE-Y : Fy force (unit: F)

ELFORCE-Z : Fz force (unit: F)

ELMOM-X : Mx moment about centroid x-axis (unit: FL)

ELMOM-Y : My moment about centroid y-axis (unit: FL)

ELMOM-Z : Mz moment about centroid z-axis (unit: FL)

ELDISP-X : Local element x displacement (element elongation for beams) (unit: L)

ELDISP-Y : Local element y displacement (zero for beam) (unit: L)

ELDISP-Z : Local element z displacement (zero for beam) (unit: L)

ELROT-X : Local x-rotation (rigid body rotation eliminated for beam) (unit: R)

ELROT-Y : Local y-rotation (rigid body rotation eliminated for beam) (unit: R)

ELROT-Z : Local z-rotation (rigid body rotation eliminated for beam)(unit: R)

ELTOR-X : torsion about centroid (unit: 1/L)

ELCUR-Y : y-curvature about centroid (unit: 1/L)

ELCUR-Z : z-curvature about centroid (unit: 1/L)

CONDIS-X : Contact element displacement in local x-direction (unit: L)

CONDIS-Y : Contact element displacement in local y-direction (unit: L)

CONDIS-Z : Contact element displacement in local z-direction (unit: L)

CONFOR-X : Contact element force in local x-direction (unit: FL^{-1} for all contact elements except CONT130, 152, 164 having unit: F).

CONFOR-Y : Contact element force in local y-direction (unit: FL^{-1} for all contact elements except CONT130, 152, 164 having unit: F).

CONFOR-Z : Contact element force in local z-direction, (unit: FL^{-1} for all contact elements except CONT130, 152, 164 having unit: F).

SUBMASS : dry mass - buoyancy mass (unit: ML^{-1})

DRYMASS : dry mass (unit: ML^{-1})

INISTRA-X : element initial axial strain (unit: -)

TEMP : element temperature (unit: ^oC)

INTPRES : internal pressure (unit: FL^{-2})

EXTPRES : external pressure (unit: FL^{-2})

INITOR-X : initial torsion (unit: L^{-1})

INICUR-Y : initial curvature about local y-axis (unit: L^{-1})

INICUR-Z : initial curvature about local z-axis (unit: L^{-1})

ICONDIS-X : Contact element initial displacement in local x-direction (unit: L)

ICONDIS-Y : Contact element initial displacement in local y-direction (unit: L)

ICONDIS-Z : Contact element initial displacement in local z-direction (unit: L)

For elements HELIX231-235 the **CONDIS** and **CONFOR** results are the bending induced effects in the contact material for the helix.

FELID: First element ID number in numerical model.

LELID: Last element ID number in numerical model.

XSCL: Scaling factor to be used for x-axis (to convert to convenient unit).

YSCL: Scaling factor to be used for y-axis (to convert to convenient unit).

ELEND: Element end 1 or 2. Default value: 1

EXAMPLE:

```
#        Raf file name      plotfilename
ELPLOT  "slay-np"           "slay-np-cdz"
#
# x-legend     x-res     y-legend                yres      felid   lelid   xscl yscl
  "Load step"  LOADSTEP  "Contact z-disp  (m)"    CONDIS-Z   2301  2316     1    1
#
#        Raf file name      plotfilename
ELPLOT  "slay-np"           "slay-np-cfz"
#
# xleg          xres     yleg                    yres      felid   lelid   xscl yscl
"Load step"    LOADSTEP  "Contact z-lload(kN/m)"  CONFOR-Z   2301   2316   1    1e3
```

3.6 GNPLOT - Global Nodal PLOTs

The **GNPLOT** format is:

GNPLOT RAFPRE MPFPRE XLEG XRES YLEG YRES FNODEID LNODEID XSCL YSCL [LOADSTEP]

RAFPRE: The BFLEX2010.raf file name prefix. If the name is defined by a combination of lower case and upper case letters, use e.g. 'input-filename'.

MPFPRE: The output .mpf file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'mpf-filename'.

XLEG: The legend name for the x-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g. 'Displacement (m)'.

XRES: x-axis result type. The following result types are available:

- **X-COR :** x-coordinate (unit: L)
- **Y-COR :** y-coordinate (unit: L)
- **Z-COR :** z-coordinate (unit: L)
- **T-COR :** Polar theta coordinate (for nodes generated by this option) (unit: R)
- **A-COR :** Polar lay angle (for nodes generated by this option) (unit: R)
- **E-COR :** curvilinear element coordinate (unit: L)
- **K-COR :** curvilinear xy-coordinate, i.e. kp-coordinate for pipeline problems requires that a seabed has been defined in BFLEX2010 (unit: L)

YLEG: The legend name for the y-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g. 'Displacement (m)'.

YRES: y-axis result type. The following result types are available:

- **X-COR :** nodal x-coordinate (unit: L)
- **Y-COR :** nodal y-coordinate (unit: L)
- **Z-COR :** nodal z-coordinate. If a seabed surface is defined (k-cor is used on x-axis) then the seabed z-coordinate will also be plotted (unit: L)
- **T-COR :** Polar theta coordinate (for nodes generated by this option) (unit: R)
- **A-COR :** Polar lay angle (for nodes generated by this option) (unit: R)
- **NODISP-X :** nodal x-displacement (unit: L)
- **NODISP-Y :** nodal y-displacement (unit: L)
- **NODISP-Z :** nodal z-displacement (unit: L)

NOROT-X : nodal x-rotations (unit: 1/L)

NOROT-Y : nodal y-rotations (unit: 1/L)

NOROT-Z : nodal z-rotations (unit: 1/L)

FNODEID: First node ID number in numerical model.

LNODEID: Last node ID number in numerical model.

XSCL: Scaling factor to be used for x-axis (to convert to convenient unit).

YSCL: Scaling factor to be used for y-axis (to convert to convenient unit). If a negative value is used, **NODISPX-Z** is transformed to local x-y system for the element, which is convenient when evaluating displacement from the route configuration for pipeline cases.

LOADSTEP: Reference load step number, only applicable for (nodisp-x), (nodisp-y) (nodisp-z) Allows the user to select which loadstep from which the nodal displacements are measured.

EXAMPLE:

```
#       transform to xy system and measure displacement from step 10
#
#       .raf prefix     .mpf prefix
GNPLOT  "slay-np"       "slay-np-xz"
#
#    Legend x       x-res.  Legend y             y-res. fnodid lnodid xscl yscl Loadstep
"X-coordinate (m)" K-COR   "KP-coordinate (m)" Z-COR  1      441    1     -1    10
```

3.7 GLPLOT - Global ELement PLOTs

The **GLPLOT** format is:

GLPLOT RAFPRE MPFPRE XLEG XRES YLEG YRES FELID LELID XSCL YSCL [ELEND INTPOI NLOADSTEP INC]

RAFPRE: The BFLEX2010.raf file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'input-filename'.

MPFPRE: The output .mpf file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'mpf-filename'.

XLEG: The legend name for the x-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g. 'Displacement (m)'.

XRES: x-axis result type. The following result types are available:

X-COR : x-coordinate (unit: L)

Y-COR : y-coordinate (unit: L)

Z-COR : z-coordinate (unit: L)

T-COR : Polar theta coordinate (for elements connected to nodes generated by this option) (unit: R)

A-COR : Polar lay angle (for elements connected to nodes generated by this option) (unit: R)

E-COR : curvilinear element coordinate (unit: L)

K-COR : curvilinear xy-coordinate, i.e. kp-coordinate for pipeline problems requires that a seabed has been defined in BFLEX2010 (unit: L)

YLEG: The legend name for the y-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g. 'Displacement (m)'.

YRES: y-axis result type. All integration point results can be applied, see Section 2.38. In addition, the following result types can be applied:

ELFORCE-X : Fx force (unit: F)

ELFORCE-Y : Fy force (unit: F)

ELFORCE-Z : Fz force (unit: F)

ELMOM-X : Mx moment about centroid x-axis (unit: FL)

ELMOM-Y : My moment about centroid y-axis (unit: FL)

ELMOM-Z : Mz moment about centroid z-axis (unit: FL)

ELDISP-X : Local element x displacement (element elongation for beams) (unit: L)

ELDISP-Y : Local element y displacement (zero for beam) (unit: L)

ELDISP-Z : Local element z displacement (zero for beam) (unit: L)

ELROT-X : Local x-rotation (rigid body rotation eliminated for beam) (unit: R)

ELROT-Y : Local y-rotation (rigid body rotation eliminated for beam) (unit: R)

ELROT-Z : Local z-rotation (rigid body rotation eliminated for beam) (unit: R)

ELTOR-X : Torsion about centroid (unit: 1/L)

ELCUR-Y : y-curvature about centroid (unit: 1/L)

ELCUR-Z : z-curvature about centroid (unit: 1/L)

CONDIS-X : Contact element displacement in local x-direction (unit: L)

CONDIS-Y : Contact element displacement in local y-direction (unit: L)

CONDIS-Z : Contact element displacement in local z-direction (unit: L)

CONFOR-X : Contact element force in local x-direction (unit: FL^{-1} for all contact elements except CONT130 having unit: F).

CONFOR-Y : Contact element force in local y-direction (unit: FL^{-1} for all contact elements except CONT130 having unit: F).

CONFOR-Z : Contact element force in local z-direction (unit: FL^{-1} for all contact elements except CONT130 having unit: F).

SUBMASS : dry mass - buoyancy mass (unit: ML^{-1})

DRYMASS : dry mass (unit: ML^{-1})

INISTRA-X : element initial axial strain (unit: -)

TEMP : element temperature (unit: ^oC)

INTPRES : internal pressure (unit: FL^{-2})

EXTPRES : external pressure (unit: FL^{-2})

INITOR-X : initial torsion (unit: L^{-1})

INICUR-Y : initial curvature about local y-axis (unit: L^{-1})

INICUR-Z : initial curvature about local z-axis (unit: L^{-1})

ICONDIS-X : Contact element initial displacement in local x-direction (unit: L)

ICONDIS-Y : Contact element initial displacement in local y-direction (unit: L)

ICONDIS-Z : Contact element initial displacement in local z-direction (unit: L)

FELID: First element ID number in numerical model.

LELID: Last element ID number in numerical model.

XSCL: Scaling factor to be used for x-axis (to convert to convenient unit).

YSCL: Scaling factor to be used for y-axis (to convert to convenient unit).

ELEND: Element end 1 or 2. Default value: 1

INTPOI: Integration point number (around cross section, point 1 start at local y=radius and z = 0 and then clockwise). If no integration point, this is the negative reference load step number. Default value: 1

NLOADSTEP: If a positive load step number is given, then the results will be measured relative to the state at that step. If no or a negative loadstep is given, then actual values are used. Default value: 0

INC: Element increment related for each nodal point. Default value: 1

EXAMPLE:

```
#       .raf prefix     .mpf prefix
GLPLOT  "slay-np"       "slay-np-ax"
#
#  Legend x    x-res. Legend y           y-res.    nodeid lnodeid xscl yscl Elend
"S-coord (m)" E-COR  "Axial force (kN)"  ELFOR-X   1      440     1     1     1
#
#       .raf prefix     .mpf prefix
GLPLOT  "slay-np"       "slay-np-sxx"
#
#  Legend x     x-res. Legend y          y-res.    nodeid lnodeid xscl yscl Elend Intpoi
"KP-coord (m)" K-COR  "Stress-xx (MPa)" Sigma-xx   1      440     1     1     1      5
```

3.8 FAPLOT- Fatigue calculation

The purpose of the **FAPLOT** option is to allow the user to perform fatigue damage calculation for all visual nodes in the structure and where the stress components xx, yy and xy are all stored. Note that this requires the following:

1. That the fatigue data relevant for structural element material is defined, see Section 2.22
2. That the result types SIGMA-XX and FATIGUE is selected, see Section 2.38

This means that it is possible to calculate the fatigue of the tensile armour directly from the .raf file without using BFPOST and LIFETIME (For the pressure armour this is still needed)

The **FAPLOT** format is:

FAPLOT RAFPRE LOFPRE I3 FTIME LTIME OPTSTR UNTCONV

RAFPRE: The .raf file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'input-filename'.

LOFPRE: The output .lof file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'fat-filename'.

I3: the number of load cycles.

FTIME: the first load step for calculating stress range.

LTIME: the last load step point for calculating stress range.

OPTSTR: Option for stress range calculation.

- $\neq$ **1 :** Stress range is taken to be the difference between the stress ranges obtained at load steps **FTIME** and **LTIME**.
- **1 :** Stress range is taken to be the largest stress range between load steps **FTIME** and **LTIME**.

UNTCONV: unit conversion factor to fit the fatigue data.

3.9 NRPLOT- Nodal Reaction PLOTs

The purpose of the **NRPLOT** option is to allow the user to sum the element forces acting on a certain node and follow this node as function of history. The total force is transformed to global system. Note that if reaction forces is to be obtained, the effect

of external loads need to be added manually. This is because only element info is stored on data base.

The **NRPLOT** format is:

NRPLOT RAFPRE MPDPRE XLEG XRES YLEG YRES FNODEID LNODEID XSCL YSCL [IEND]

RAFPRE: The BFLEX2010.raf file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'input-filename'.

MPFPRE: The output .mpf file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'mpf-filename'.

XLEG: The legend name for the x-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g. 'Displacement (m)'.

XRES: x-axis result type. The following result types are available:

LOADSTEP : Load step

TIME : Time

HISTN : History number with reference to BFLEX2010 input file, e.g. HIST100

YLEG: The legend name for the y-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g. 'Displacement (m)'.

YRES: y-axis result type. The following result types are available:

QX-REACT : Fx restoring force (unit: F)

QY-REACT : Fy restoring force (unit: F)

QZ-REACT : Fz restoring force (unit: F)

MX-REACT : Mx restoring moment about centroid x-axis (unit: FL)

MY-REACT : My restoring moment about centroid y-axis (unit: FL)

MZ-REACT : Mz restoring moment about centroid z-axis (unit: FL)

QX-DAMP : Fx damping force (unit: F)

QY-DAMP : Fy damping force (unit: F)

QZ-DAMP : Fz damping force (unit: F)

MX-DAMP : Mx damping moment about centroid x-axis (unit: FL)

MY-DAMP : My damping moment about centroid y-axis (unit: FL)

MZ-DAMP : Mz damping moment about centroid z-axis (unit: FL)

QX-INER : Fx inertia force (unit: F)

QY-INER : Fy inertia force (unit: F)

QZ-INER : Fz inertia force (unit: F)

MX-INER : Mx inertia moment about centroid x-axis (unit: FL)

MY-INER : My inertia moment about centroid y-axis (unit: FL)

MZ-INER : Mz inertia moment about centroid z-axis (unit: FL)

QX-ALL : Fx restoring+damping+inertia force (unit: F)

QY-ALL : Fy restoring+damping+inertia force (unit: F)

QZ-ALL : Fz restoring+damping+inertia force (unit: F)

MX-ALL : Mx restoring+damping+inertia moment about centroid x-axis (unit: FL)

MY-ALL : My restoring+damping+inertia moment about centroid y-axis (unit: FL)

MZ-ALL : Mz restoring+damping+inertia moment about centroid z-axis (unit: FL)

FNODEID: First node ID number in numerical model.

LNODEID: Last node ID number in numerical model.

XSCL: Scaling factor to be used for x-axis (to convert to convenient unit).

YSCL: Scaling factor to be used for y-axis (to convert to convenient unit).

IEND: The element end that shall contribute. Default value = 0 means that all element ends are considered.

3.10 RSPLOT - ReStart PLOT

The ReStart PLOT format is:

RSPLOT RAFPRE MPFPRE LOADSTEP FNODEID LNDOEID NODESCL VALSCL

RAFPRE: The .raf file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'input-filename'.

MPFPRE: The output file name If the file name is defined by a combination of lower case and upper case letters, use e.g. 'mpf-filename'.

LOADSTEP: Load step number.

FNODID: First node ID number in numerical model.

LNODEID: Last node ID number in numerical model.

NODESCL: Scaling factor to be used for node number (i = i - NODESCL).

VALSCL: Scaling factor to be used for x-,y and z-values (xi = xi * VALSCL) .

The output file format is:

Node number, Xi , Yi , Zi , T11, T12, T13, T21, T22, T23, T31, T32, T33

3.11 ENPLOT- ENvelope PLOTs

The ENvelope PLOT format is:

ENPLOT RAFPRE MPFPRE XRES XSCL YSCL

RAFPRE: The BFLEX2010.raf file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'input-filename'.

MPFPRE: The output .mpf file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'mpf-filename'.

XRES: The x-axis type, that may have the following value:

E-COR : Curvilinear s-coordinate

ELEM : Element or node ID number

XSCL: Scaling factor to be used for x-axis (to convert to convenient unit).

YSCL: Scaling factor to be used for y-axis (to convert to convenient unit).

Then all defined envelopes will be plotted stored on .mpf files.

EXAMPLE:

```
#       raf file     plot file       xres  xscl  yscl
ENPLOT  "slay-np"   "envelopes-np"   ELEM  1     1
```

3.12 BFPOST - BFlex POSTprocessing

The purpose of this command is to create a raf file that enables activating the modules BOUNDARY and PFLEX for creating a pressure armour model for a flexible pipe and to calculate the fatigue damage by LIFETIME for a given (arbitrary) section of pipe. This requiers that a flexible cross-section is used to generate the structural model, see Section 2.10.

Note that both the carcass and pressure spiral must be described using a **BFLEX** geometry, see Section 2.9. I.e. the **CCODE** parameter in **CROSSECTION FLEXCROSS** (see Section 2.10) can not be set to **MANUAL** if the model is to be further processed by BOUNDARY and PFLEX.

The **BFPOST** format is:

BFPOST RAFPRE IEL1 IEL2 NCYCLE PFLEXMESH

RAFPRE: The BFLEX2010.raf file name prefix. If the name is defined by a combination of lower case and upper case letters, use e.g. 'input-filename'

IEL1: First core element

IEL2: Last core element

NCYCLE: Number of cycles the load history in the bif-file represents

PFLEXMESH: Mesh factor for PFLEX model. Default = 1

EXAMPLE:

```
#       rafpre   iel1    iel2  ncycle  pflexmesh
bfpost itcode0-3    1      20       1          1
#
```

BFLEX2010POST will by using the above input, generate a file named RAFPRE_BFLEX2010.raf. This file provides the necessary info for PFLEX, BOUNDARY and LIFETIME.

3.13 GRPLOT - ELement Reaction PLOTs

The purpose of the GRPLOT feature is to allow summing up all element reactions belong to the same element end node and to plot these along the given axis. The **GRPLOT** format is:

GRPLOT RAFPRE MPFPRE XLEG XRES YLEG YRES FELID LELID XSCL YSCL [ELEND NLOADSTEP ITRANS]

RAFPRE: The BFLEX2010.raf file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'input-filename'.

MPFPRE: The output .mpf file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'mpf-filename'.

XLEG: The legend name for the x-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g. 'Displacement (m)'.

XRES: x-axis result type. The following result types are available:

X-COR : x-coordinate (unit: L)

Y-COR : y-coordinate (unit: L)

Z-COR : z-coordinate (unit: L)

E-COR : curvilinear element coordinate (unit: L)

K-COR : curvilinear xy-coordinate, i.e. kp-coordinate for pipeline problems requires that a seabed has been defined in BFLEX2010 (unit: L)

YLEG: The legend name for the y-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g. 'Displacement (m)'.

YRES: y-axis result type. All integration point results can be applied, see Section 2.38. In addition, the following result types can be applied:

QX-REACT : Fx force (unit: F)

QY-REACT : Fy force (unit: F)

QZ-REACT : Fz force (unit: F)

MX-REACT : Mx moment about centroid x-axis (unit: FL)

MY-REACT : My moment about centroid y-axis (unit: FL)

MZ-REACT : Mz moment about centroid z-axis (unit: FL)

FELID: First element ID number in numerical model.

LELID: Last element ID number in numerical model.

XSCL: Scaling factor to be used for x-axis (to convert to convenient unit).

YSCL: Scaling factor to be used for y-axis (to convert to convenient unit).

ELEND: Element end 1 or 2. The macimum value is 2. Default value: 2

NLOADSTEP: If a positive load step number is given, then the results will be measured relative to the state at that step. If no or a negative loadstep is given, then actual values are used. Default value: 0

ITRANS: If the value 1 is given the element end reactions are transformed to global system. Default value: 0

EXAMPLE:

```
#       .raf prefix     .mpf prefix
GRPLOT  "slay-np"       "slay-np-ax"
#
#  Legend x    x-res. Legend y           y-res.    nodeid lnodeid xscl yscl Elend
"S-coord (m)" E-COR  "Axial force (kN)"  QX-REACT   1      440     1     1     1
#
```

4 DYNPOSTB Report Generator

4.1 General

The **DYNRES** results are stored on .dyn files having the same prefix as the BFLEX2010 input files. Each time series plot is stored in consecutive order and may be plotted from the XPOST program. The DYNPOSTB program is applied for generation if ASCII files for each time series. As opposed to BFLEX2010POST, the results are available for all time steps.

DYNPOSTB requires an input file with extension .bdi and will report **MXPLOT** on a log file ending with extension .bdo and **DYNPLOT** to .mpf files specified by the user. All output is given on MATRIXPLOT (.mpf) format, which are ASCII files that can be further processed by e.g. Matlab or Python scripts.

When running DYNPOSTB from the command line, the basis is to write the name referring to DYNPOSTB in the prompt. In addition, the following optional command line arguments can be applied:

- Adding -m and a float number representing the multiplication factor for scaling of the default local work memory size used by DYNPOSTB.
- Adding -n and the input file name prefix makes DYNPOSTB execute the post-processing without further requesting the user to specify the file name.
- Adding -d and a float number representing the multiplication factor for scaling of the default memory size for retrieving **DYNRES** results from BFLEX2010.

The optional arguments above may be specified in an arbitrary order. The option -d may be required when BFLEX2010 also has been executed with the -d option as described in Section 1.7.

EXAMPLE:

```
command line: DYNPOSTB  -n prefixname  -m 3.4  -d 7.2
```

4.2 Input data to DYNPOSTB

All input is described on ASCII file format. The maximum line length is 136 characters and the maximum number of letters in one single text string is 32.

Comment text strings are defined by introducing # at the start of the line.

The following identifiers define the different data groups:

- **MXPLOT** for MaXima or MiNima postprocessing.
- **DYNPLOT** for dynamic plot postprocessing.

In the following the input data will be explained.

4.2.1 MXPLOT - MaXima/MiNima postprocessing

The format is:

MXPLOT I1 R2 I3 I4 R5 A6 A7 - AN

I1 : Extreme value type:

- **1 :** Find maxima
- **-1 :** Finds minima
- **2 :** Finds difference from local maxima to following local minima. By local the extreme value between two crossings of the mean value in the current result file is meant.
- **-2 :** Finds difference from local minima to following local maxima. By local the extreme value between two crossings of the mean value in the current result file is meant.

R2 : The fraction relative to the absolute maxima/minima found that will be evaluated

I3 : The **DYNRES** number in consecutive order as defined in the simla input file

I4 : number of maxima to be counted

R5 : start time of evaluation (to avoid transcient)

A6 : output file name prefix

A7-AN : prefix of all dyn files to be processed

EXAMPLE:

```
#       imax fraction idynres nmaxim tstart outputfile .dyn files
MXPLOT     1 0.3      1       100    30.0   "test"     "screening_270_9"
                                                      "screening_181_9"
```

```
                                                    "screening_225_9"
```

Typical result file:

EXAMPLE:

```
Nodal   Velocity      Node     1001 Dof  1  Maxima Statistics
   time                value         seastatefile
  0.3490E+02  0.1477E+00  screening_181_9.dyn
  0.8490E+02  0.1227E+00  screening_181_9.dyn
  0.7740E+02  0.9049E-01  screening_181_9.dyn
  0.6420E+02  0.7781E-01  screening_181_9.dyn
  0.9290E+02  0.7075E-01  screening_181_9.dyn
  0.5390E+02  0.2325E+00  screening_225_9.dyn
  0.9670E+02  0.1657E+00  screening_225_9.dyn
  0.6760E+02  0.8834E-01  screening_225_9.dyn
  0.7380E+02  0.8607E-01  screening_225_9.dyn
```

4.2.2 DYNPLOT - Dynamic plot postprocessing

The format is:

DYNPLOT A1 A2 I3 [I3B] R4

A1 : the .dyn file name

A2 : the .mpf plot storage file name

I3 : the plot number

I3B : optional parameter. If specified, the plot number range i3-i3b will be stored to the same .mpf file

R4 : y-axis scaling factor

EXAMPLE:

```
#        raf file    plot file        plotno   y-scale
DYNPLOT  "slay-nt"   "slay-nt-my"       1      1e-3
```

EXAMPLE:

```
#        raf file    plot file        plotnos   plotnoe y-scale
DYNPLOT  "slay-nt"   "slay-nt-my"       1         3     1e-3
```

5 PFLEX Input Guide

The PFLEX input options are given on the prompt or using the command line option:

EXAMPLE:

```
command line: PFLEX -n prefix
```

The program will take *prefix.pif* as the input file. The content of a typical *prefix.pif* file is:

```
NFILES
N
CHRAF       INODE ILOCAL [DELTA0 MOMCTR IPROC]
RAF_1       INODE_1 ILOCAL_1 DELTA_1  MOMCTR_1 IPROC_1
...
RAF_I       INODE_I ILOCAL_I DELTA_I  MOMCTR_I IPROC_I
...
RAF_N       INODE_N ILOCAL_N DELTA_N  MOMCTR_N IPROC_N
```

where

N Number of analysis.

CHRAF The prefix of the 2bif file.

INODE Node position for the PFLEX model.

ILOCAL Local deformation parameter.

DELTA0 Initial ovalization, *optional*.

MOMCTR Bending moment scaling parameter, *optional*.

IPROC Procedure parameter, *optional*.

For the command line option the input are the following:

File The prefix of the bif file.

Node position for the PFLEX model The global node number (INOD) where the PFLEX local model is positioned (typically the node having the largest stress from

the BFLEX2010 analysis), see also Section 2.1. By this parameter it is possible for the user to study various positions along the BFLEX2010 model without having to perform a new run by BFLEX2010.

Local deformation parameter If the local deformation parameter (ILOCAL) = 1, only local deformations are included during visualization, else global rigid body deformations will be included as well.

Initial ovalization Delta0 Optional, default value = 0.002.

Bending moment scaling parameter MOMCTR Optional, default value = 0.0. The squeeze pressure from bending times curvature is factorized with MOMCTR. Setting the parameter to 1.0 will switch this effect on.

Procedure parameter IPROC Optional, default value = 2. Setting the IPROC parameter to 0 will activate the new analytical model of Dec 2016 having the following features:

- The new model combines solving the membrane force by applying the curved beam model as before, however, by applying an analytical model to find the normal curvature.
- The model considers the directionality of the load. Hence, the fatigue damage will be distributed differently
- The low pressure stability problem noted by several users has been fixed.
- The spike problem noted by NOV on summer 2016 has been fixed.
- The Carcass and pressure spiral are working together in one model.
- Gives a very good fit to NOV ovalisation test data (see separate memo) if MOMCTR is set to 0.

Applying the IPROC value of 1 in combination with MOMCTR value 1.0 means that the old model from before February 2015 is applied having the following features:

- Including the the tension-curvature term (Bugg fix of Fall 2012)
- The support condition spring scaling parameter is set to 1e6 for all support springs.
- The membrane force and the normal curvature is found by applying the curved beam model supported by springs with a stiffness scaled by the above factor.

- The model does not consider the directionality of the load i.e. the load always acts in the same plane and direction.
- The Carcass and pressure spiral are treated separately.
- Problems have been encountered with respect to numerical stability for low pressure cases as for the NOV ovalisation test cases.
- The spike problem noted by NOV on summer 2016 has been fixed.

Usinf the IPROC default value of 2 gives the model of February 2015 (Bflex 3.09. official version) with the following features:

- Including the the tension-curvature term
- The support condition spring scaling parameter is set to 1e4 and a scaling factor of $\cos\theta^{40}$ is applied to concentrate the support constraint to obtain better fit with NOV ovalisation test data.
- The membrane force and the normal curvature is found by applying the curved beam model supported by springs with a stiffness scaled by the above factors.
- The model does not consider the directionality of the load i.e. the load always acts in the same plane and direction.
- The Carcass and pressure spiral are treated separately.
- The spike problem noted by NOV on summer 2016 has been fixed.
- Gives a very good fit to NOV ovalisation test data if MOMCTR is set to 0. If MOMCTR is set to 1, the ovalisation was observed to be underestimated for the high tension cases. This is caused by the two deformation modes related to tension-curvature and moment-curvature disturbing each other when the moment-curvature term starts to play a role.

Based on the calibration studies carried out, the following alternives are recommeded for future use of the Pflex program.

- Either, use the default model (Bflex 309 of Summer 2016) i.e. IPROC=2, in combination with MOMCTR = 0 for high tension cases, and MOMCTR=1 for small tension cases.
- or, use the new analytical model i.e. IPROC=0, in combination with MOMCTR = 0 for high tension cases, and MOMCTR=1 for small tension cases.

6 BOUNDARY input guide

Note: BOUNDARY must be executed before PFLEX.

The BOUNDARY input options are given on the prompt or using the command line option:

EXAMPLE:

```
command line: BOUNDARY -n prefix
```

The program will take *prefix.boi* as the input file. The content of a typical *prefix.boi* file is:

```
NFILES
N
CHRAF INODE FIMOD ILOCAL ISTRES NONLIN ISTPFR ISTPBE ICODE TRCURP
RAF_1   INODE_1 FIMOD_1 ILOCAL_1 ISTRES_1 NONLIN_1 ISTPFR_1 ISTPBE_1 ICODE_1 TRCURP_1
...
RAF_I   INODE_I FIMOD_I ILOCAL_I ISTRES_I NONLIN_I ISTPFR_I ISTPBE_I ICODE_I TRCURP_I
...
RAF_N   INODE_N FIMOD_N ILOCAL_N ISTRES_N NONLIN_N ISTPFR_N ISTPBE_N ICODE_N TRCURP_N
```

The meaning of each parameter can be referred to Chapter 6.

For the BOUNDARY command line option the input are the following:

File The prefix of the bif file.

Node position for the PFLEX model The global node number (INOD) where the PFLEX local model is positioned (typically the node having the largest stress from the BFLEX analysis), see also Section 2.1.

Orientation of boundary model Angular position of BOUNDARY model (FIMOD) according to Fig. 2.14 (unit: deg), see also Section 2.1. By the INOD and FIMOD parameters it is possible for the user to study various positions along the BFLEX model without having to perform a new run by BFLEX.

Local deformation parameter If the local deformation parameter (ILOCAL) = 1, only local deformations are included during visualization, else global rigid body

deformations will be included as well.

Local stress parameter If the local stress parameter (ISTRES) = 1, the boundary stress analysis is carried out for all bodies in the Boundary model. If ISTRES = 0 no transverse stress analysis is performed, If ISTRES = 2, the stress analysis is carried out for the mid body only.

Non-linear analysis parameter If the Non-linear analysis parameter (NONLIN) = 1, then residual stress analysis is carried out using the non-linear material properties; else the analysis will be elastic.

Load step for friction application The load step from which friction is applied (ISTPFR)

Load step for bending application The load step from which bending of the pipe starts (ISTPBE)

Icode ICODE = 1 Calculate the stress inside the cross sectional area. This option is time consuming.

ICODE ≠ 1 The stress inside the cross sectional area is not calculated.

Transverse curvature parameter The transverse curvature parameter is used to avoid zero stiffness in the transverse direction (TRCURP). The default value is zero.[1].

[1]The value is given as a fraction of the normal curvature

7 LIFETIME input guide

LIFETIME requires the life time data to be stored on the model.lif file. The format of the model.lif file is as described in Section 7.1.

LIFETIME can be run from command line without any arguments, in which case the user is promted both for the prefix of the life time file and the output file where results are to be stored. Alternatively LIFETIME can be started with the following option:

- Adding -n and the life time file prefix makes LIFETIME execute without further requesting the user to specify the file name. In this case the output file is required to be supplied in the *.lif file

EXAMPLE:

```
command line: LIFETIME  -n prefixname
```

Two **additional** lines are then required at the start of the the *.lif file:

```
CHOUT
PREFIX.LOF
NFILES
N
CHRAF       CHINCL NCYCLE NSTART NEND IFLAG
RAF_1       CHINCL_1 NCYCLE_1 NSTART_1 NEND_1 IFLAG_1
...
RAF_I       CHINCL_I NCYCLE_I NSTART_I NEND_I IFLAG_I
...
RAF_N       CHINCL_N NCYCLE_N NSTART_N NEND_N IFLAG_N
```

where the second line sould be the result file name, full name needs to be specified. The meaning of other parameters can be referred to Section 7.1.

7.1 Format of the life time data (.lif file)

The life time file *.lif, when supplied interactively at the prompt

NFILES
N
FILE **INCLUDE** **NCYCLE** **NSTART** **NEND** **IFLAG**
PREFIX$_1$ **YORN**$_1$ **NCYC**$_1$ **NS**$_1$ **NE**$_1$ **IFL**$_1$
...
PREFIX$_I$ **YORN**$_I$ **NCYC**$_I$ **NS**$_I$ **NE**$_I$ **IFL**$_I$
...
PREFIX$_N$ **YORN**$_N$ **NCYC**$_N$ **NS**$_N$ **NE**$_N$ **IFL**$_N$

where

file The prefix of the raf file being part of the analysis.

include A code specifying whether the .raf file is to be included or not. Allowable values are **Y**(es) or **N**(o).

ncycle Number of cycles corresponding to each model.raf file.

nstart Start load step for calculating the stress range.

nend End load step for calculating the stress range.

iflag Options for stress sange calculation

$\neq$ **1** Stress range is taken to be the difference between the stress ranges at **NSTART** and **NEND**.

$=$ **1** Stress range is taken to be the largest stress range between **NSTART** and **NEND**.

EXAMPLE:

```
#--------------------------------------------------------------------
# LIFETIME
#--------------------------------------------------------------------
NFILES
6
FILE        INCLUDE        NCYCLE            NSTART          NEND   IFLAG
  near1       Y          91790304              25            45      1
  near3       Y            563135              40           100      1
  near4       Y             80151              60           160      1
  near5       Y             13123              80           260      1
  near6       Y              1153             130           360      1
#
```

8 BPOST Report Generator

8.1 General

BFLEX2010POST 3.3.1 includes the following files:

prefix.bpi the BFLEX2010POST Input File.

prefix.bpl the BFLEX2010POST Log File where warnings and error messages are written.

In addition references are given in the input file to the relevant .raf file and the associated output .mpf files. All output is given on MatrixPlot (.mpf) format.

8.2 Input data

All input is described on ASCII file format. The maximum line length is 136 characters and the maximum number of letters in one single text string is 32.

Comment text strings are defined by introducing # at the start of the line.

The following identifiers define the different data groups:

NOPLOT for NOdal history PLOTs

NOPLO2 for NOdal history PLOts 2

ELPLOT for ELement history PLOTs

GNPLOT for Global Nodal PLOTs

GLPLOT for Global Element PLOTs

In the following the input data will be explained.

8.3 NOPLOT - NOdal history PLOTs

The **NOPLOT** format is:

NOPLOT RAFPRE MPFPRE XLEG XRES YLEG YRES FNODID LNODID XSCL YSCL

RAFPRE: The BFLEX2010POST raf-file name prefix. If the name is defined by a combination of lower case and upper case letters, use e.g. 'input-filename'

MPFPRE: The output .mpf file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'mpf-filename'

XLEG: The legend name for the x-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g 'Displacement (m)'

XRES: x-axis result type. The following result types are available:

LOADSTEP : Load step

HISTN : History number with reference to BFLEX2010 input file, e.g HIST100

YLEG: The legend name for the y-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g. 'Displacement (m)'

YRES: y-axis result type. The result types available:

SIGMA-AX : Axial stress σ_{xa} (unit: MPa)

SIGMA-NX : Normal curvature stress σ_{xbn} (unit: MPa)

SIGMA-TX : Transverse curvature stress σ_{xbt} (unit: MPa)

SIGMA-L1 : Total longitudinal stress corner 1 of tensile armour, i.e. the sum of the above σ_{xx} (unit: MPa).

SIGMA-L2 : Total longitudinal stress corner 2 of tensile armour, i.e. the sum of the above σ_{xx} (unit: MPa).

SIGMA-L3 : Total longitudinal stress corner 3 of tensile armour, i.e. the sum of the above σ_{xx} (unit: MPa).

SIGMA-XX : Total longitudinal stress σ_{xx} (unit: MPa)

SIGMA-YY : Transverse stress σ_{yy} (unit: MPa)

SIGMA-ZZ : Normal stress σ_{zz} (unit: MPa)

SIGMA-YZ : Shear stress σ_{zy} (unit: MPa)

CONT-PRE : Contact pressure (unit: MPa)

REL-DISP : Relative displacement (unit: mm)

HEL-TORS : Helix torsion (unit: mm-1)

HEL-NCUR : Helix normal curvature (unit: mm-1)

HEL-TCUR : Helix transverse curvature (unit: mm-1)

FNODEID: First node ID number in visual model

LNODEID: Last node ID number in visual model

XSCL: Scaling factor to be used for x-axis (to convert to convenient unit)

YSCL: Scaling factor to be used for y-axis (to convert to convenient unit)

EXAMPLE:

```
#
# nodal plot
#       .raf prefix     .mpf prefix  Legend x
NOPLOT  "test1"      "test-nosax"    "Prescribed rotation (deg)"
#  x-res.     Legend y                y-res.      Node 1   Node 2 X-fac Y-fac
 HIST400   "Axial stress (MPa)"   SIGMA-AX    5264     5265   1     1
#       .raf prefix     .mpf prefix   Legend x
NOPLOT  "test1"       "test-nosl1"   "Prescribed rotation (deg)"
#x-res.     Legend y               y-res.      Node 1   Node 2 X-fac Y-fac
HIST400   "Long str c1  (MPa)"   SIGMA-L1    5264     5265   1     1
#
```

8.4 NOPLO2 - NOdal history PLOts 2

To enable to present the visual model node result along a strip of pipe with increasing node order and curvilinear cooordinate. This is practical when performing 3D heat analysis using *e.g.* ABAQUS and where input in terms of heat generation from friction work (BFLEX2010) is needed. **NOPLO2** can also be used to plot ovalization of the carcass.

The **NOPLO2** format is:

NOPLO2 RAFPRE MPFPRE XLEG XRES YLEG YRES INO1 INO2 XSCAL YSCAL STEP0 INC

RAFPRE: The BFLEX2010POST raf-file name prefix. If the name is defined by a combination of lower case and upper case letters, use e.g. 'input-filename'

MPFPRE: The output .mpf file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'mpf-filename'

XLEG: The legend name for the x-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g 'Displacement (m)'

XRES: x-axis result type. The following result types are available:

E-COR : Load step

YLEG: The legend name for the y-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g. 'Displacement (m)'

YRES: y-axis result type. The result types available:

NODISP-X : Nodal displacement in x-direction (global system) (unit: mm)

NODISP-Y : Nodal displacement in y-direction (global system) (unit: mm)

NODISP-Z : Nodal displacement in z-direction (global system) (unit: mm)

SIGMA-AX : Axial stress σ_{xa} (unit: MPa)

SIGMA-NX : Normal curvature stress σ_{xbn} (unit: MPa)

SIGMA-TX : Transverse curvature stress σ_{xbt} (unit: MPa)

SIGMA-L1 : Total longitudinal stress corner 1 of tensile armour, i.e. the sum of the above σ_{xx} (unit: MPa).

SIGMA-L2 : Total longitudinal stress corner 2 of tensile armour, i.e. the sum of the above σ_{xx} (unit: MPa).

SIGMA-L3 : Total longitudinal stress corner 3 of tensile armour, i.e. the sum of the above σ_{xx} (unit: MPa).

SIGMA-XX : Total longitudinal stress σ_{xx} (unit: MPa)

SIGMA-YY : Transverse stress σ_{yy} (unit: MPa)

SIGMA-ZZ : Normal stress σ_{zz} (unit: MPa)

SIGMA-YZ : Shear stress σ_{zy} (unit: MPa)

CONT-PRE : Contact pressure (unit: MPa)

REL-DISP : Relative displacement (unit: mm)

HEL-TORS : Helix torsion (unit: mm^{-1})

HEL-NCUR : Helix normal curvature (unit: mm^{-1})

HEL-TCUR : Helix transverse curvature (unit: mm^{-1})

ENERGY : Energy (unit: MJ)

INO1: First node ID number in visual model

INO2: Last node ID number in visual model

XSCL: Scaling factor to be used for x-axis (to convert to convenient unit)

YSCL: Scaling factor to be used for y-axis (to convert to convenient unit)

ISTEP0: All values in **YRES** is subtracted by the value at load step **ISTEP0**

INC: Node increment

EXAMPLE:

```
#
# nodal plot 2
#       .raf prefix     .mpf prefix   Legend x   x-res   Legend y      y-res
NOPLO2  "q4"            "q4-l1-90"    "e-cor"    e-cor  "Heat (J/m^2)" energy
#
# ino1  ino2  xscale yscale     istep0 inc
  9355  10507   1     1         130     16
#
```

8.5 ELPLOT - ELement history PLOTs

The **ELPLOT** format is:

ELPLOT RAFPRE MPFPRE XLEG XRES YLEG XRES FELID LELID ELEND XSCL YSCL

RAFPRE: The BFLEX2010POST raf-file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'input-filename'.

MPFPRE: The output .mpf file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'mpf-filename'.

XLEG: The legend name for the x-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g. 'Displacement (m)'.

XRES: x-axis result type. The following result types are available:

LOADSTEP : Load step

HISTN : History number with reference to BFLEX2010 input file, e.g. HIST100

YLEG: The legend name for the y-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g. 'Displacement (m)'.

YRES: y-axis result type. The following result types are available:

ELFORCE-X : F_x force (unit: N)

ELFORCE-Y : F_y force (unit: N)

ELFORCE-Z : F_z force (unit: N)

ELMOM-X : M_x moment about centroid x-axis (unit: Nmm)

ELMOM-Y : M_y moment about centroid y-axis (unit: Nmm)

ELMOM-Z : M_z moment about centroid z-axis (unit: Nmm)

ELTOR-X : Torsion about centroid (unit: 1/mm)

ELCUR-Y : y-curvature about centroid (unit: 1/mm)

ELCUR-Z : z-curvature about centroid (unit: 1/mm)

Note: The following result types apply for the last BFLEX2010 element in the segment with respect to tension, internal pressure, external pressure and torsion moment. The **FELID**, **LELID** and **ELEND** parameters are dummy parameters as the results are given for all layers.

LA_AXFO : Axial force per layer

LA_TOMO : Torsion moment per layer

LA_PI : Pressure at inside of each layer + outside at last layer

LA_GAP : Gap between each interface

LA_DR : Radial displacmeent of each layer

LA_DT : Thickness change per layer

LA_SXX : Longitudinal stress per layer (along helix for helices)

LA_SZZ : Normal (radial) stress per layer

FELID: First element ID number in numerical model.

LELID: Last element ID number in numerical model.

ELEND: Element end (1 or 2)

XSCL: Scaling factor to be used for x-axis (to convert to convenient unit).

YSCL: Scaling factor to be used for y-axis (to convert to convenient unit).

EXAMPLE:

```
#
# element history plot
#       .raf prefix .mpf prefix   Legend x
ELPLOT  "test1"    "test-elfx"    "Prescribed rotation (deg)"
#x-res.    Legend y               y-res.      el 1   el 2  end X-fac Y-fac
 HIST400   "Axial force x (kN)"   ELFORC-X    10     10    1   1     1e-3
#
#       .raf prefix .mpf prefix   Legend x
ELPLOT "test1"    "test-elfy"    "Prescribed rotation (deg)"
#x-res.    Legend y              y-res.      el 1   el 2  end X-fac Y-fac
HIST400   "Shear force y (kN)"   ELFORC-Y    10     10    1   1     1e-3
#
#       .raf prefix .mpf prefix   Legend x
ELPLOT  "test1"    "test-elfz"    "Prescribed rotation (deg)"
#x-res.    Legend y              y-res.      el 1   el 2  end X-fac Y-fac
HIST400   "Shear force z (kN)"   ELFORC-Z    10     10    1   1     1e-3
#
#
```

8.6 GNPLOT - Global Nodal PLOTs

The **GNPLOT** format is:

GNPLOT RAFPRE MPFPRE XLEG XRES YLEG XRES FNODEID LNODEID XSCL YSCL

RAFPRE: The BFLEX2010POST raf-file name prefix. If the name is defined by a combination of lower case and upper case letters, use e.g. 'input-filename'.

MPFPRE: The output .mpf file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'mpf-filename'.

XLEG: The legend name for the x-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g. 'Displacement (m)'.

XRES: x-axis result type. The following result types are available:

X-COR : x-coordinate (unit: mm)
Y-COR : y-coordinate (unit: mm)
Z-COR : z-coordinate (unit: mm)
E-COR : curvilinear element coordinate (unit: mm)

YLEG: The legend name for the y-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g. 'Displacement (m)'.

YRES: y-axis result type. The following result types are available:

X-COR : nodal x-coordinate (unit: mm)
Y-COR : nodal y-coordinate (unit: mm)
Z-COR : nodal z-coordinate (unit: mm)
NODISP-X : nodal x-displacement (unit: mm)
NODISP-Y : nodal y-displacement (unit: mm)
NODISP-Z : nodal z-displacement (unit: mm)
NOROT-X : nodal x-rotations (unit: mm)
NOROT-Y : nodal y-rotations (unit: mm)
NOROT-Z : nodal z-rotations (unit: mm)

FNODEID: First node ID number in numerical model.

LNODEID: Last node ID number in numerical model.

XSCL: Scaling factor to be used for x-axis (to convert to convenient unit).

YSCL: Scaling factor to be used for y-axis (to convert to convenient unit).

EXAMPLE:

```
# global nodal plot
#       .raf prefix  .mpf prefix      Legend x
GNPLOT  "test1"      "test-gnodisx"   "Length (mm)"
#x-res.    Legend y         y-res.    Node 1   Node 2 X-fac Y-fac
 E-COR     "X-displ (mm)"   NODISP-X  1        81     1     1
#
#       .raf prefix  .mpf prefix      Legend x
GNPLOT  "test1"      "test-gnodisz"  "Length (mm)"
#x-res.    Legend y         y-res.    Node 1   Node 2 X-fac Y-fac
 E-COR     "Z-displ (mm)"   NODISP-Z  1        81     1     1
```

8.7 GLPLOT - Global ELement PLOTs

The **GLPLOT** format is:

GLPLOT RAFPRE MPFPRE XLEG XRES YLEG XRES FNODEID LNODEID ELEND XSCL YSCL

RAFPRE: The BFLEX2010POST raf-file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'input-filename'.

MPFPRE: The output .mpf file name prefix. If the file name is defined by a combination of lower case and upper case letters, use e.g. 'mpf-filename'.

XLEG: The legend name for the x-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g. 'Displacement (m)'.

XRES: x-axis result type. The following result types are available:

X-COR : x-coordinate (unit: mm)

Y-COR : y-coordinate (unit: mm)

Z-COR : z-coordinate (unit: mm)

E-COR : curvilinear element coordinate (unit: mm)

YLEG: The legend name for the y-axis. If the legend name contains more than one word or if the name is defined by a combination of lower case and upper case letters, use e.g. 'Displacement (m)'.

YRES: y-axis result type. All the result types defined as integration point results. In addition, the following result types can be applied:

ELFORCE-X : F_x force (unit: N)

ELFORCE-Y : F_y force (unit: N)

ELFORCE-Z : F_z force (unit: N)

ELMOM-X : M_x moment about centroid x-axis (unit: Nmm)

ELMOM-Y : M_y moment about centroid y-axis (unit: Nmm)

ELMOM-Z : M_z moment about centroid z-axis (unit: Nmm)

ELTOR-X : torsion about centroid (unit: 1/mm)

ELCUR-Y : y-curvature about centroid (unit: 1/mm)

ELCUR-Z : z-curvature about centroid (unit: 1/mm)

CONFOR-Z : Contact element force in local z-direction (surface normal) (unit: N).

CONDIS-Z : Contact element displacement in local z-direction (surface normal) (unit: mm).

FNODEID: First element ID number in numerical model.

LNODEID: Last element ID number in numerical model.

ELEND: Element end 1 or 2.

XSCL: Scaling factor to be used for x-axis (to convert to convenient unit).

YSCL: Scaling factor to be used for y-axis (to convert to convenient unit).

EXAMPLE:

```
#       .raf prefix     .mpf prefix Legend x
GLPLOT  "test1"       "test-glfx"   "Length (mm)"
#
# x-res.    Legend y              y-res.      El1   El2  End    X-fac Y-fac
E-COR     "Axial force  (kN)" ELFORC-X        1     80    1      1      1e-3
#
```

9 Examples

9.1 General remarks

In the following, FE analyses examples will be presented, investigating different aspects of flexible pipe and cable structural behaviour. These analyses are based on application of the finite elements included in BFLEX2010. The examples presented in the following include:

- 2D cross-section stress analysis of a 4 inch Wellstream flexible pipe
- 2D & 3D stress behaviour of flexible riser at the platform hang-off by
- Lateral buckling behaviour of tensile armours by 3D modelling
- Lateral buckling by Repeated Unit Cell modelling (harmonic boundary conditions)

9.2 2D cross-section stress analysis of a 4 inch flexible pipes

9.2.1 Objectives

As noted in Section 1.8 most models used to predict stresses due to bending of helical structures are based on the assumption that no end effects are present. This means that the stresses at one section is uniquely determined by the kinematic quantities in terms of the axial strain, torsion and curvature at that section with no consideration of the helix as a 3D structure that will transfer forces along the helix path whenever transients are present. This assumption enables the use of stress resultant based models that are typically based on the bi-linear moment-curvature behaviour noted for flexible pipes in combination with elastic models to describe axial and torsion stiffness.

The purpose of the present example is to explore the alternative 2D modelling approaches mentioned in Section 1.8, both with regard to axisymmetric stresses, moment-curvature behaviour and the resulting friction stresses.

Table 9.1: CROSS-SECTION PARAMETERS - 4 INCH RISER ID = 101.6 MM

No	Description	Layer thickness (mm)	Material	Dimension	Number of wires	Lay angle (Deg)
1	Carcass	6.4	Steel1	$36mm^2$	1	87.828
2	Liner	5.1	Plastic1	-	-	-
3	Z-spiral	6.4	Steel2	$79.23mm^2$	1	87.813
4	Antiwear1	2.0	Plastic2	-	-	-
5	Tensile1	2.0	Steel3	$2 \times 5mm$	61	-38
6	Antiwear	2.0	Plastic2	-	-	-
7	Tensile2	2.0	Steel3	2×5 mm	65	38
8	Sheath1	0.3	Plastic3	-	-	-
9	Sheath2	6.0	Plastic3	-	-	-

Table 9.2: MATERIAL PROPERTIES - 4 INCH RISER

Material	Young's modulus (MPa)	Poisson's ratio (-)
Steel1	$2.0 \cdot 10^5$	0.30
Steel2	$1.96 \cdot 10^5$	0.30
Steel3	$2.07 \cdot 10^5$	0.30
Plastic1	$1.1 \cdot 10^3$	0.33
Plastic2	$1.1 \cdot 10^3$	0.35
Plastic3	$1.1 \cdot 10^3$	0.40

9.2.2 Input data

The 4 inch Wellstream pipe tested in 1990 as part of the FPS2000 project has been used as basis (Skallerud, 1991). The test pipe was first exposed to 20 MPa internal pressure, then followed by cyclic curvature at $+/-0.053m^{-1}$ where the moment-curvature relation was recorded. No detailed information that relates to the plastic materials were available. These properties were therefore assumed and tuned against the slip bending stiffness, basically representing all plastic layers by same Young's modulus. However, for the antiwear tapes, the friction coefficient between tensile armour and tape and the representative shear modulus relevant for shear interaction behaviour have been obtained by small scale testing, see (Dai et al., 2017) and respectively obtained as $\mu = 0.22$ and $G = 221MPa$. The cross-section data and mechanical properties are presented in Table 9.1 and Table 9.2.

9.2.3 Modelling

The modelling alternatives with case labels are defined in Table 9.3 where the following abbreviations has been applied:

- RUC means repeated unit cell modelling. Applying the inner tensile armour as a reference the pitch length is divided by the number of wires. The model includes two pitches, one on each side of the mid node 2 of alltogether 3 centre nodes. A master slave equation is introduce between relevant dofs having same circumferential coordinate. Then to obtain symmetry, the outer tensile armour lay angle had to be adjusted to 37.8 deg in order to obtain the same pitch length/number of wire ratio.
- MCM refers to moment-curvature resultant based modelling. Cases not included MCM are based on the SBM approach.
- ITC refers to the **ITCODE** procedure applying the CAFLEX model to obtain the axisymmetric stresses which was the original procedure applied by BFLEX.
- SHM refers to applying the shear interaction effect based on the measured shear modulus of the tape. In order to calculate the shear stiffness, the measured shear modulus of $221MPa$ was applied. Then an effective thickness of $5.1+2=7.1mm$ was assumed for the inside interface of the inner tensile armour whereas $2mm$ was assumed between the tensile armours.
- P52 refers to the element type **PIPE52**
- NF refers to appliation of the new friction model with constant stick stiffness.
- HC and HS refer to **HCONT** and **HSHEAR** element series.
- FAC menas a reduction factor applied on the measured fricion factor as explained below.

The case labels include the element types involved, whether or not shear interaction is included and whether the new friction model (with constant stick stiffness) or the old friction model (with variable stick stiffness) are applied.

All modelling alternatives are based on flexible pipe automatic cross-section modelling procedures inherent in BFLEX2010 except for Alternative 5 which is based on manual modelling by using the FRICONTACT contact material feature for the interface between tubular and helix. For the case of opposite lay angles of same magnitude as in this case, the effective friction factor given above is reduced by a factor $\cos\alpha$ considering the direction of the slip vector.

Table 9.3: ALTERNATIVE MODELLING PROCEDURES - 4 INCH RISER

No	Case label	Number of elements
1	MCM-ITC21-P52	6
2	MCM-ITC31-P52	6
3	MCM-ITC32-P52	6
4	MCM-ITC32-P52-SHM	6
5	MCM-HC464-HS364-Manually modelled	6
6	RUC-ITC0-P52-HS352	254
7	RUC-ITC1-P52-HS352	254
8	RUC-ITC1-P52-HS352-NF	254
9	RUC-HC464-HC463-HS464-HS353	656
10	RUC-HC464-HC463-HS464-HS353-NF	656
11	RUC-HC464-HC453-HS464-HS353-NF	526
12	RUC-HC464-HC453-HS464-HS353-NF-SHM	526
13	RUC-HC464-HC463-HS464-HS353-NF-SHM-FAC	526

The RUC and **ITCODE** models both included two segments, i.e. 3 centre nodes. In the former case, the pipe length is governed by the pitch length versus number of wires ratio, whereas in the latter case a length of 20 mm was applied. This was also the case for the HS364-HC464 alternative, however then only 2 nodes were applied.

The pipe was first exposed to internal pressure and thereafter prescribed rotation was introduced to cause bending throughout one cycle in one plane.

9.2.4 Results in terms of axisymmetric stresses

Fig. 9.1 compares the results obtained from different modelling procedures with respect to the through thickness stress σ_{zz} at 20 MPa internal pressure. It is reasonable to believe that the CAFLEX Lagrange multiplier procedure (all the **ITCODE** procedures) would yield the most accurate results as compared to the penalty method applied for the new frameworks developed, the latter being based on selecting the radial stiffness to be 10 times $\frac{E}{t}$ for the softest interface. However, it is found that the deviation in σ_{zz} is within 10 % for all layers. Fig. 9.2 compares the corresponding results for the axial wire stress σ_{xx}, where the deviation is found to be even smaller and within 6 % for all layers.

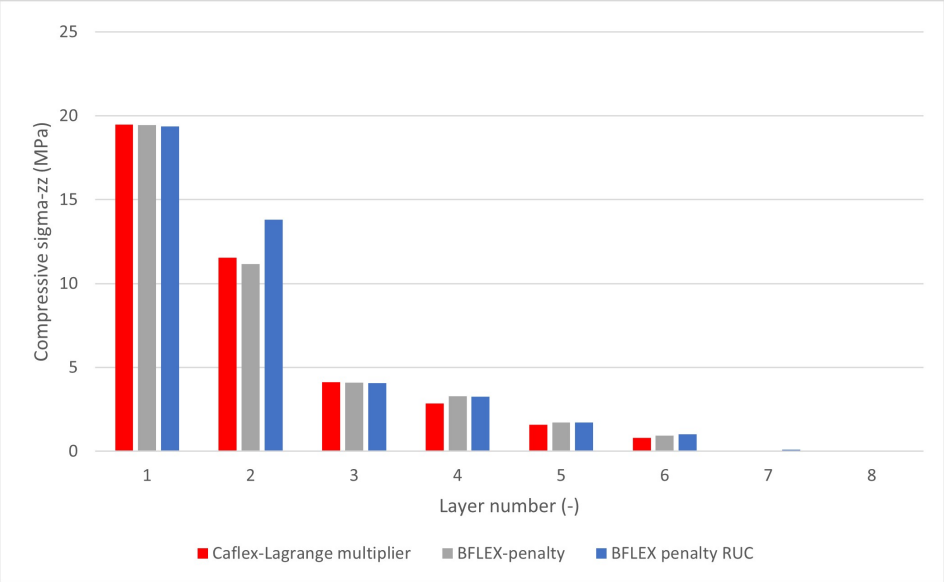


Figure 9.1: Comparing σ_{zz} with respect to different contact algorithm at internal pressure 20MPa

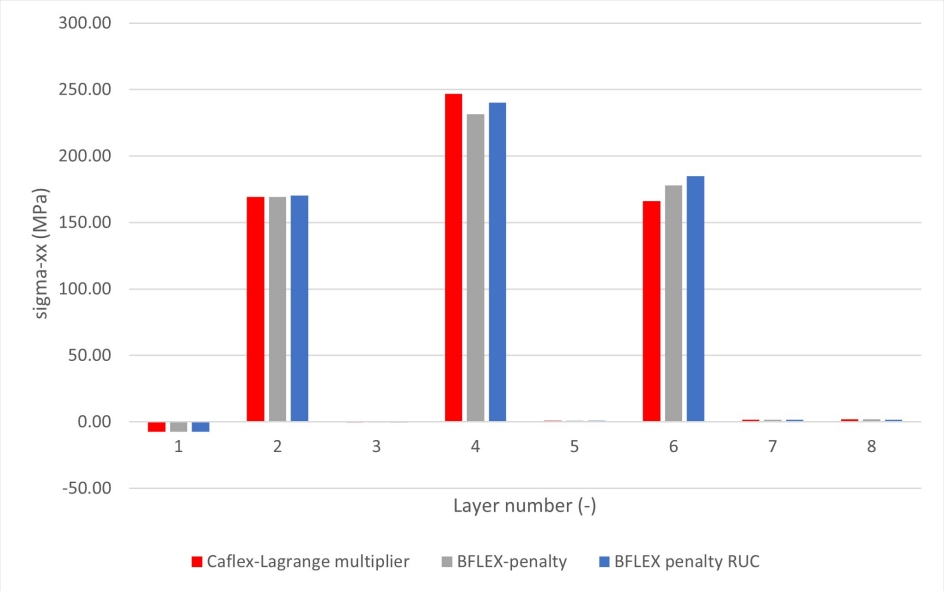


Figure 9.2: Comparing σ_{xx} with respect to different contact algorithm at internal pressure 20MPa

9.2.5 Results in terms of bending moment and friction stresses

Fig. 9.3 and Fig. 9.4 compares the results obtained by the different modelling procedures listed in Table 9.3 with respect to the measured moment curvature behaviour. As expected and when the default parameters are applied, there are almost now differences between the old and new friction models. It is also seen that several of the algorithms have a tendency of overestimating the energy dissipation during one cycle as compared

to the experiment. The total energy dissipation best fits are obtained by MCM-ITC31-P52, RUC-HC464-HC453-HS364-HS353, RUC-HC464-HC453-HS364-HS353-NF-SHM and RUC-HC464-HC463-HS364-HS353-NF-SHM-FAC. This is because the **HCONT453** element considers contact between the two tensile layers directly with no consideration of the antiwear layer. This means that the slip vector will be governed by the two armour layer motions which will result in a friction line load with a factor $\cos\alpha$ smaller than obtained by allocating two sets of **HCONT463** contact elements on each side of the same antiwear layer, which would then treat the armour motions as decoupled. This can be compensated by modifying the applied friction factor at the two interfaces by a factor 0.8 as shown for RUC-HC464-HC463-HS364-HS353-NF-SHM-FAC. It is also noted that **ITCODE31** decribes this effect quite well, the reason being that it was calibrated against the fibre optic Bragg data, see (Sævik, 2011). By looking into the stick-slip region in more detail with reference to Fig. 9.4 it is seen that the best fit is obtained for the SHM cases.

Fig. 9.5 and Fig. 9.6 show the respective inner armour layer friction stresses at the outer fibre location of the pipe. Taking the RUC-HC464-HC453-HS364-HS353-NF-SHM and RUC-HC464-HC463-HS364-HS353-NF-SHM-FAC cases as the reference since they gave a best fit for the moment curvature diagram, it is seen that the **ITCODE21** procedure underestimates the stresses whereas the **ITCODE31** is on the conservative side. Good fit with respect to slope is also obtained by the **ITCODE32** procedure when combined with including the shear interaction effect. It is seen that the slip value is overestimated. This can be improved by scaling the friction factor for the two interfaces between the tensile armour layers, as noted above. No scaling or shear interaction parameters should be included when applying **ITCODE31** method because it has already been calibratied against full scale test data.

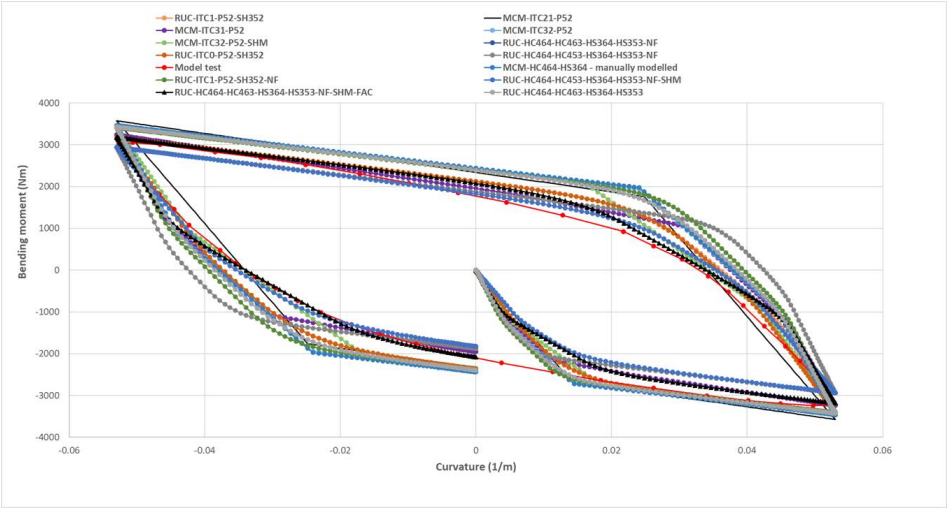


Figure 9.3: Comparing moment curvature obtained by alternative modelling procedures against test data

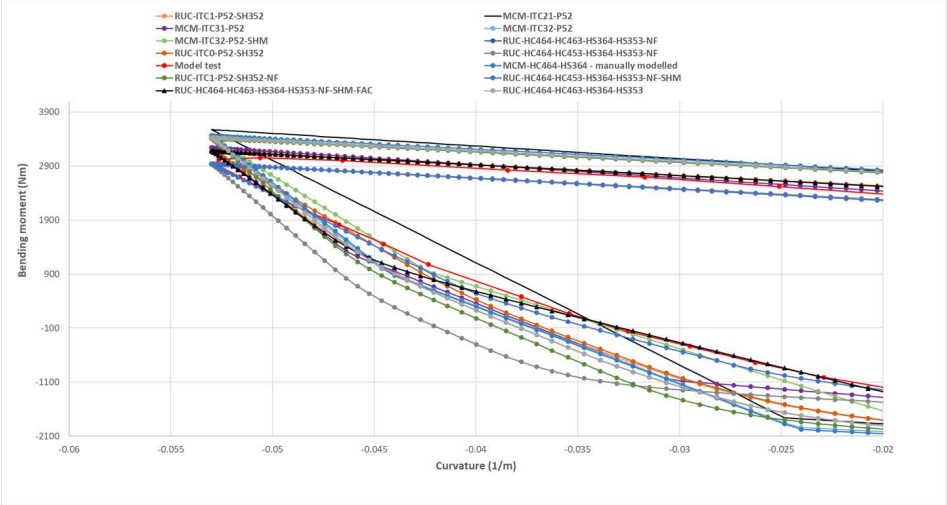


Figure 9.4: Zoom of moment curvature obtained by alternative modelling procedures versus test data

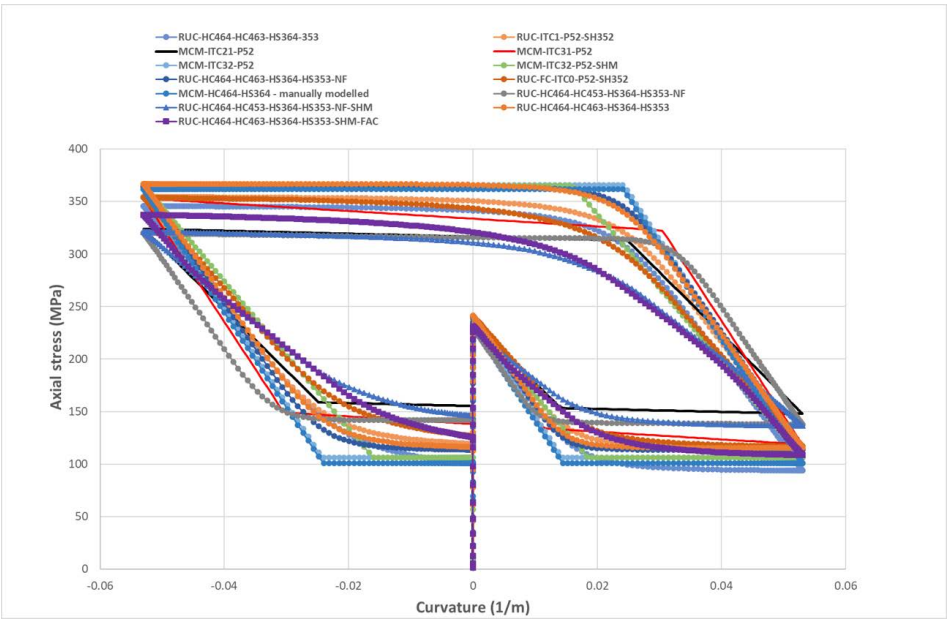


Figure 9.5: The corresponding axial friction wire stress at outer fibre in inner armour during one curvature cycle

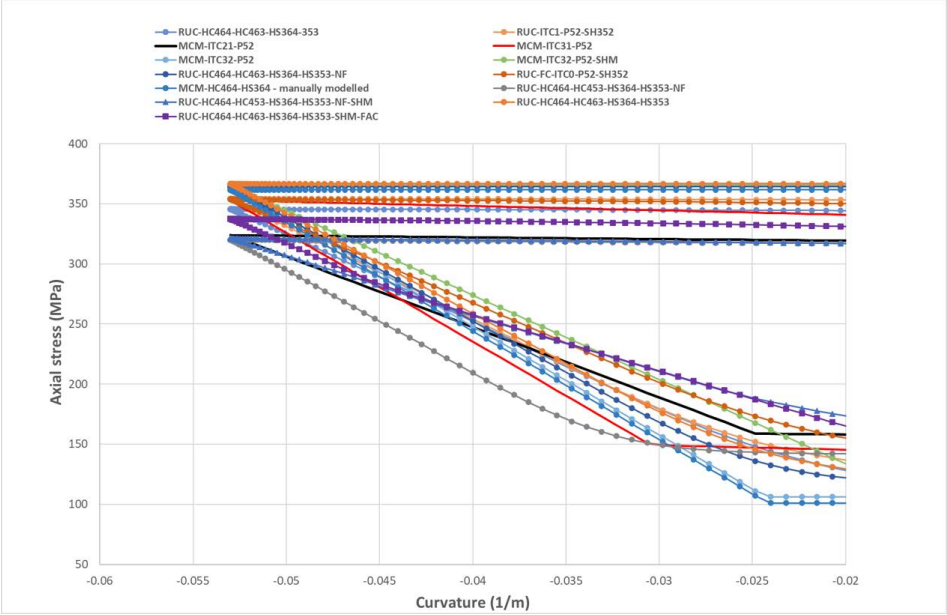


Figure 9.6: Zoom of the corresponding axial friction wire stress at outer fibre in inner armour during one curvature cycle

9.3 Stress behaviour of flexible riser at the platform hang-off

9.3.1 Objectives

As noted above most models used to predict stresses due to bending of helical structures are based on the assumption that no end effects are present. This means that the stresses at one section is uniquely determined by the kinematic quantities in terms of the axial strain, torsion and curvature at that section with no consideration of the helix as a 3D structure that will transfer forces along the helix path. This assumption enables the use of stress resultant based models that are based on the bi-linear moment-curvature behaviour noted for flexible pipes.

In the present example the task is to see if this will be the case for a pipe where the bending curvature is applied at the end fitting of the flexible pipe. This will be done by applying two different models: one based on the 2D **PIPE52** moment formulation and one based on the 3D **HSHEAR352** helix formulation, hereafter referred to as the moment and helix models.

9.3.2 Input data

The case selected represented the upper hang-off section of a flexible riser. In order to control the curvature at the the hang-off section, a bend stiffener is in most cases applied on the outside of the pipe, a conical cylinder normally made of polyurethane. The model included the end fitting where the tensile armour is terminated by epoxy moulding. The flexible pipe section was 10 m long whereas the bend stiffener section was 5 m long, see Fig. 9.7. Other input data are found in Table 9.4.

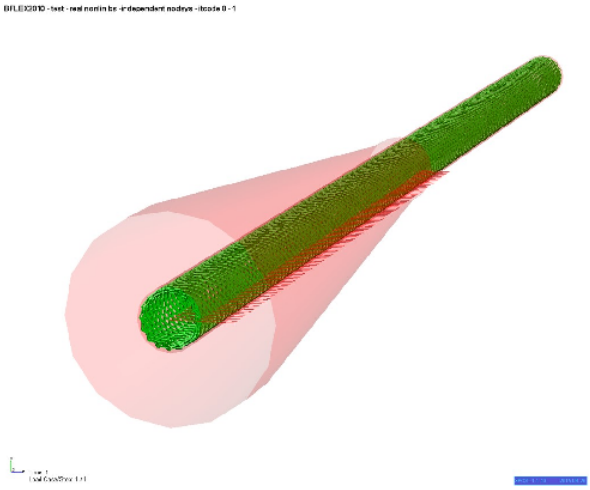


Figure 9.7: Model applied to simulate tensile armour stresses at bend stiffener section of flexible riser

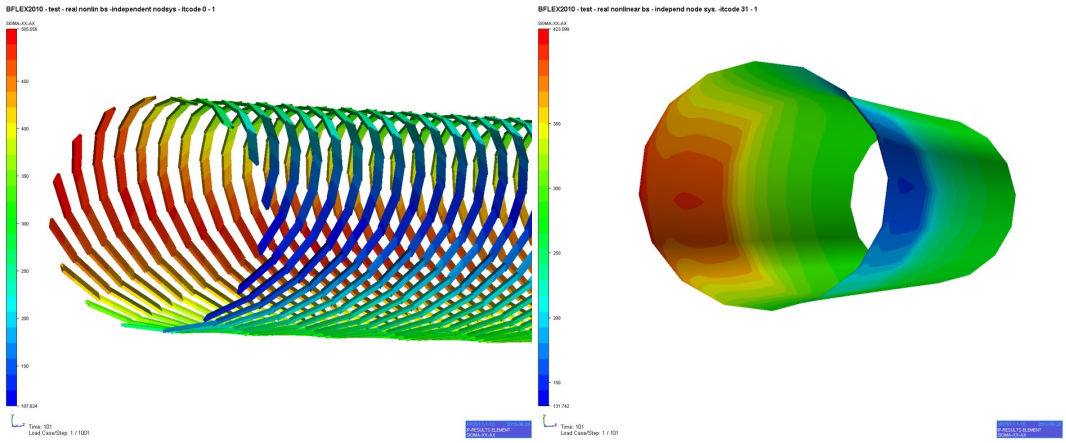
Table 9.4: INPUT PARAMETERS - SIMULATION OF FLEXIBLE RISER

Parameter	**Value**	**Unit**
Inside radius	108.8	mm
Pressure spiral area	8.6	mm^2/mm
Tensile wire area	47.6	mm^2
Helix radius - inner tensile	135	mm
Number of wires - inner tensile	44	
Lay angle - inner tensile	-40	deg
Helix radius - tensile 2	141	mm
Lay angle - tensile 2	40	deg
Number of wires - tensile 2	46	
Helix radius - tensile 3	145.5	mm
Lay angle - tensile 3	-40	deg
Number of wires - tensile 3	48	
Helix radius - tensile 4	151.7	mm
Lay angle - tensile 4	40	deg
Number of wires - tensile 4	50	
Friction coefficient	0.10	
Bend stiffener (BS) inner radius	180	mm
BS maximum diameter	1096	mm
BS minimum diameter	428	mm
Radial gap between pipe and BS	12	mm
Young's modulus BS	150	MPa
Contact stiffness	500	MPa
Applied tension	1000	kN
Internal pressure	20	MPa
Applied end angle at BS end	+/-3	deg

9.3.3 Modelling

The pipe was modelled as a fixed-pinned beam with prescribed rotations at the fixed end. The pinned end was free to translate in the longitudinal global direction where a tension of 1000 kN was applied. The prescribed rotation was used to simulate the relative rotation between the riser and the floater, whereas the tension represented the weight of the riser. The overall layout of the model is shown in Fig. 9.7.

The pipe was divided into 200 equally long segments which gives an element length of 50 mm. The model included seven groups: the core, the inner tensile layer, then tensile layers 2-4, the bending stiffener and contact elements to simulate contact between the pipe and the bend stiffener.



(a) Stress distribution - helix model (b) Stress distribution - moment model

Figure 9.8: Stress distribution at end fitting for alternative helix FE models

For the moment model, all layers were modelled by **PIPE52** elements whereas for the helix model the tensile armour layers were modelled by **HSHEAR352** elements. For the helix model, the tensile armour was fixed against translation in the longitudinal helix direction to account for the end fitting fixation. Contact was handled by CONT130 pipe in pipe contact elements. A 16 point resolution was applied in the circumferential direction.

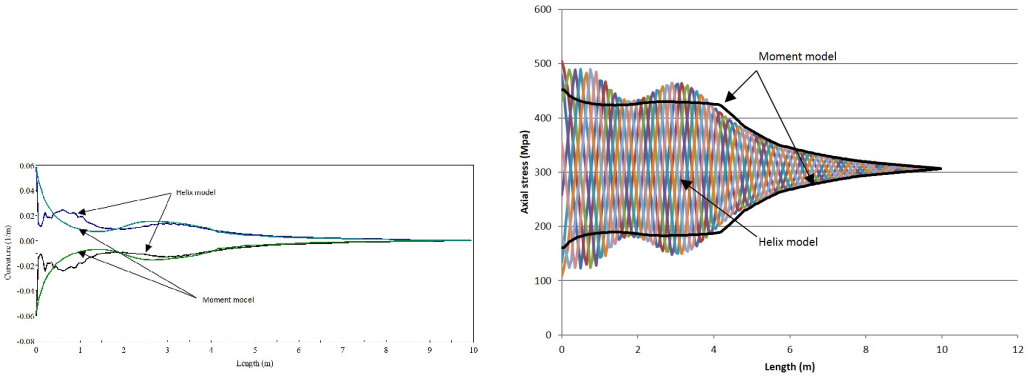
The pipe was first exposed to internal pressure and tension and thereafter prescribed rotation was introduced to cause bending throughout two cycles in one plane.

9.3.4 Results

Fig. 9.8 shows the axial stress distribution in the inner tensile armour at the end fitting at maximum end rotation of the last cycle. For the moment model, the stress distribution is symmetric whereas for the helix model, significant stresses also occur at the neutral axis. This is due to the transient effect introduced by the fact that the helix is fixed at the end fitting as noted by Lutchansky (Lutchansky, 1969).

By further comparing the associated curvature distributions at maximum and minimum end rotations during the last cycle resulting from the two models in Fig. 9.9 (a), this is also manifested by a transient in the curvature distribution of the helix model whereas the moment model poses smooth and continuous behaviour. However, by moving away from the end fitting, the two models converge against each other. This is a result of the transient wire slip behaviour resulting from the end fitting constraint.

Looking at the associated axial stress distributions along the different helices in Fig. 9.9 (b), the black line represents the maximum axial stresses obtained from the moment



(a) Comparison of curvature distributions (b) Comparison of axial stress distributions

Figure 9.9: Comparison of curvature and stress distributions using alternative helix FE models

model. It is seen that the stresses from the helix model start out from a larger value then converging against the moment model value. This is caused by two effects: firstly by the end fitting restraint effect noted above and secondly by the non-constant curvature distribution causing un-symmetric sliding of the wires.

It is therefore clear that a stress concentration factor should be included when applying moment models when the end fitting restraint is close to the curved section. In practice, however, the end fitting is normally placed away from the curved section. Experience has shown that if this distance is longer than 0.5-1 helix pitch length, the moment model will give results that are similar to the ones obtained by a helix model.

9.4 Lateral buckling behaviour of tensile armours

9.4.1 Objectives

The objective of this example is to investigate the correlation between FE analysis and available test data (Østergaard, 2012) related to the lateral buckling behaviour of tensile armour in flexible pipe cross-sections exposed to a compressive force in the ends and then exposed to cyclic bending to a near constant curvature in one direction. Part of this is also a proposed buckling criteria to be applied in the FE analysis which is based on first yield in the tensile armour which is the same approach as used in the Timoshenko equation for tubular collapse, see also (Zhou et al., 2015).

9.4.2 Input data

Three different flexible pipe cross-sections were tested by Østergaard (Østergaard, 2012) based on applying a constant compressive force to the tensile armour in combination with cyclic bending. No external pressure was applied, thus the experiments reflected the assumption of wet annulus conditions. Primary variables in the tests were the curvature ranges and the pretension level of the inner pipe (the end cap force). The major test principle was based on mounting an approximately 6 m long flexible pipe specimen into a test bench with one end fixed in torsion whereas the other end was free to rotate. Another pipe was mounted on the inside. The inner pipe was pre-tensioned by a hydraulic actuator to simulate the end cap external overpressure (deep water pipes) and the ends were rotated by cycles to introduce curvature variations until pipe failure. The pipe curvature varied between a near zero curvature to a predefined value thus simulating the pipe motion at the touch down point during installation (or operation in dynamic applications). The length variation and the end rotation was measured and the failure was observed as severe torsion deformation of the pipe. The characteristics of the test pipes are summarized in Table 9.5.

The specimens were exposed to a constant compressive axial force and a cyclic curvature history between straight (zero curvature) and fully curved conditions as specified in Table 9.6.

9.4.3 Modelling

The model is shown in Fig. 9.10 and included the pretension pipe, the layers inside the tensile layers, two tensile layers, one structural tape layer between the two tensile layers, one anti-buckling layer and one sheath layer. The pipe was divided into 200 segments giving a helix element length of approximately $35mm$ for all cases. The pretension pipe was modelled by the elastic pipe element **PIPE31**. The other layers inside the tensile armour were modelled by layers of **HSHEAR363** elements connected to the same node system in the layer direction. The inner and outer tensile layers were represented by **HSHEAR353** elements, allowing for bi-directional slip. Each tensile layer was represented by four wires. The structural tape, anti-buckling tape and the sheath were modelled by layers of **HSHEAR363** elements. Contact between layers were handled by **HCONT453** elements. The friction between the layers was represented by applying a Coulomb model in the contact elements with a friction factor of 0.15 and an associated stick displacement of $0.1mm$. The anti-buckling tape and the sheath were merged into the same node system, which means that there were four interface contact layers: inside inner tensile - core, outside inner tensile - structural tape, outside structural tape - inside inner tensile layer and finally outside outer tensile layer - anti-buckling tape.

The left end was fixed in translation and torsion rotation whereas the right end was free to move in the horizontal direction and to rotate in torsion.

There were two loads applied in the model: cyclic bending about the global Y-axis and compression by pretension in the centre pipe. The analysis of the first 10 seconds was static, and the next 2460-4010 s (depending on case) was made dynamic. Firstly, an initial axial strain was applied to the centre pipe that was given a small axial stiffness to compensate for length variation during the failure process. Then the prescribed end rotations were applied from 10 s to 2460/4010 s, represented by a ramping function from zero to the specified curvature. Each bending cycle lasted 20 s and the applied time increment was 0.005 s for the dynamic analysis.

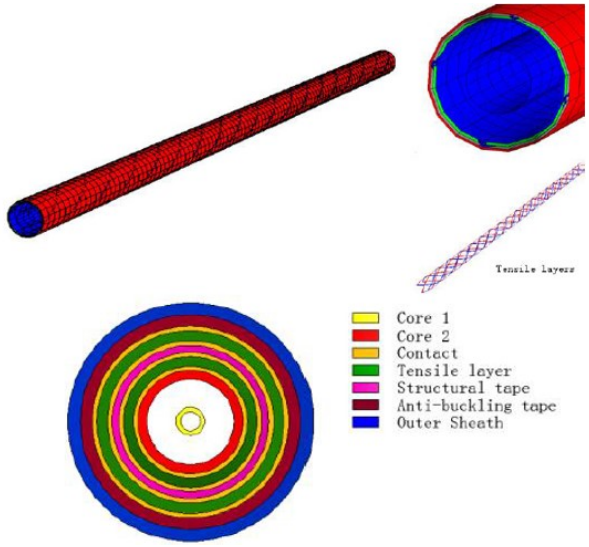


Figure 9.10: Model for tensile armour lateral buckling

9.4.4 Results

The instrumentation applied during the test included measuring the shortening of the specimen and the associated torsion rotation of the free end (Østergaard, 2012). Failure was assumed when severe torsion rotation of the specimen had occurred. This was then followed by dissecting the specimen. Failure/No failure was finally judged due to the general state of the tensile armour, whether or not the armour was severely disorganised or plastically deformed such that it would affect the pipe's ability to withstand further loading. The testing included a number of cycles in the range 18-6000 whereas a realistic number of cycles from a computation point of view is in the range of 200 cycles. The failure criteria applied during the computations was therefore stated as: if the stress in the tensile armour exceeded the yield stress during 200 cycles, then the specimen was considered to fail. The stress measure included the sum of components as shown in Fig. 9.11 were measured at the model midpoint.

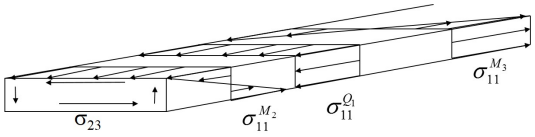


Figure 9.11: Components of stress

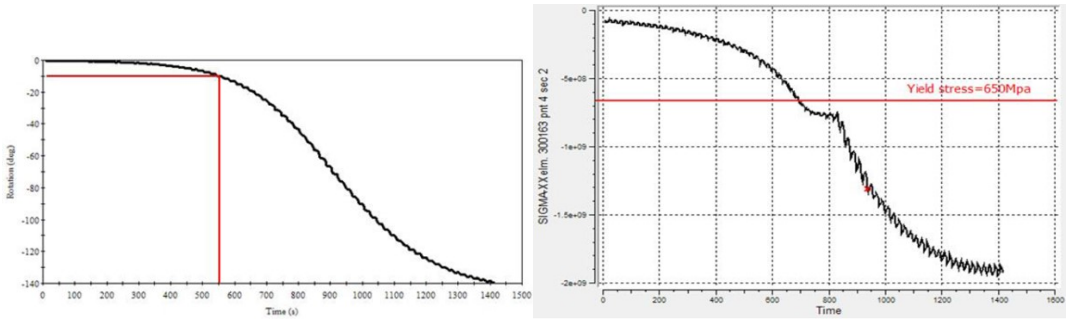
The test and numerical results in terms of failure/no failure and end rotation are presented in Table 9.7. It is seen that good correlation is obtained with respect to failure/no failure between the tests and the numerical model, using the yield stress failure criteria, the numerical results, however, on the conservative side for three cases. Differences are also noted with respect to the end rotation. This is as expected because there are obvious uncertainties with respect to the parameters applied in the model e.g. the friction coefficient and the stick-slip parameter. Also the tensile armour element is based on elastic material behaviour and is therefore not valid beyond yield. The failure mechanism results from the compressive axial load where the two tensile armour layers will be squeezed against the anti-buckling tape creating a gap between the tensile armour layers and the other layers. If the anti-buckling tape is sufficiently strong to prevent radial buckling, the wire has only one way to move, and that is sideways. The friction forces available to resist buckling is smaller for the inner layer, hence the inner layer will loose its axial load capacity first. The buckling process will therefore be initiated at the inner layer. As the axial load capacity is reached in the inner layer, this must be compensated by a loss in the axial compression forces in the outer layer as well, to keep the cross-section in torque balance. Hence the pipe must rotate in the same direction as the lay angle of the outer layer. This corresponds to a negative rotation at the right end of the model for all cases since the outer tensile lay angle is negative for all test pipes. This is confirmed in Fig. 9.12 and Fig. 9.13 which show the torsion rotation and stress versus time for a failure and no-failure case, respectively. The red line in Fig. 9.12 indicates that the yield stress is reached.

A graphical representation of the failure mechanism and the pipe configuration in a failed pipe model are shown in Fig. 9.14. It is seen that severe global torsion deformation occurs during failure.

9.5 Lateral buckling behaviour of tensile armours taking lateral contact into account

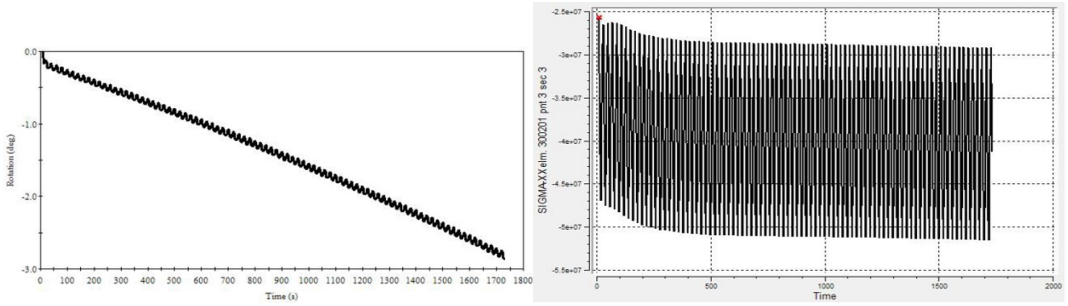
9.5.1 Objectives

The objective of this example is to investigate the role of lateral contact with respect to lateral buckling behaviour. The fill factor of the tensile armour layers for the test



(a) End torsion rotation versus time (b) Stress versus time

Figure 9.12: Development of torsion and stress for a tensile armour buckling - failure load case



(a) End torsion rotation versus time (b) Stress versus time

Figure 9.13: Development of torsion and stress for a tensile armour buckling - no failure load case

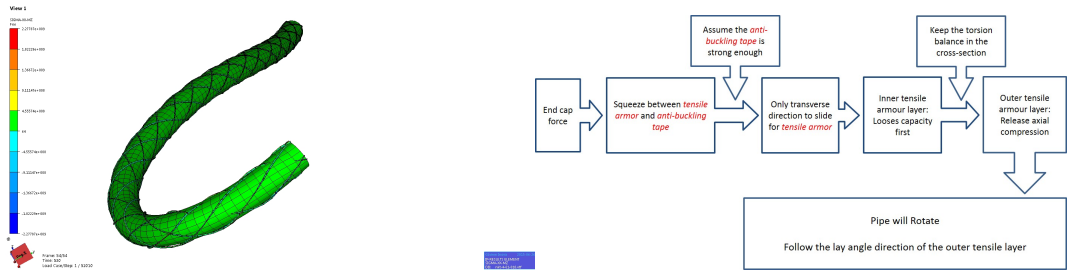
pipes investigated by Østergaard (Østergaard, 2012) was about 0.9. However, as the lateral buckling phenomenon is associated with gap closure and openings in the lateral direction, the fill factor of tensile armour in deep water pipes is normally optimized to take benefit from the stabilizing effect associated with the buckling process before plastification takes place. In order to capture such effects, lateral contact between armour wires need to be considered.

9.5.2 Input data

The same flexible pipe cross-section investigated by Lukassen (Lukassen, 2019) was investigated, see Table 9.8.

9.5.3 Modelling

The model is shown in Fig. 9.15. The model was based on assuming constant curvatures and no ovalisation, which allows for applying the Repeated Unit Cell (RUC) approach



(a) Pipe configuration at the final stage (b) Process of tensile armour buckling

Figure 9.14: The tensile armour buckling process

as described by (Lukassen et al., 2019). This means that one pitch away from each side of a defined symmetry point is divided into as many cells as there are wires in the tensile layer being focused on, which in this case means the inner tensile armour layer. Then by utilizing that the motion is only depending on the angular position, each cell can be repositioned into the line of symmetry, making a short stub model, extending one cell length on each side of the symmetry point (one cell can of course be divided into more elements if desired). Each cell have separate nodes, where the nodes positioned at the symmetry point act as masters and the end nodes having same angular position act as slaves defined by describing adequate kinematic constraints. In BFLEX2010, this is straight forward because the helix dofs are defined in the local curvilinear system (there is no need for updating the constraint coefficients).

One problem of this approach is that the cell length projection along the centre axis of the outer armour need to be the same as for the inner layer. The lay angle of the outer tensile armour was therefore slightly adjusted. The layers inside the tensile armour were modelled by **HSHEAR363** elements connected to the same node system in the layer direction. The inner and outer tensile layers were represented by **HSHEAR353** elements in separate node systems for each angular segment. The anti-wear tape, anti-buckling tape and the sheath were modelled by layers of **HSHEAR363** elements. Contact between layers were handled by **HCONT463** elements, whereas the lateral contact effect was modelled by **HCONT454**, (Dai et al., 2020) which include the extrados and intrados gap kinematics. The friction between the layers was represented by applying a Coulomb model in the contact elements with a friction factor of 0.12 and a constant stick stiffness, reflecting plane surfaces remain plane conditions (Dai et al., 2018).

The load condition was based on a constant reversed end-cap load of 1200 kN combined with a curvature cycling between 0 and $0.06m^{-1}$. The reversed end-cap effect was obtained by a prescribed pretension to elastic pipe elements at the centre and by associating a low axial stiffness to these elements **PIPE31**. The end cap load was applied statically, whereas the curvature was applied dynamically applying a period of $20s$ periods with a time step of $0.1s$

The midpoint centrenode was fixed in all directions.

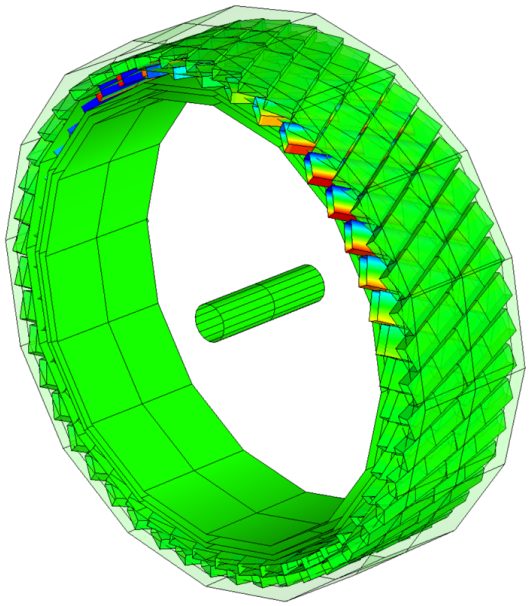
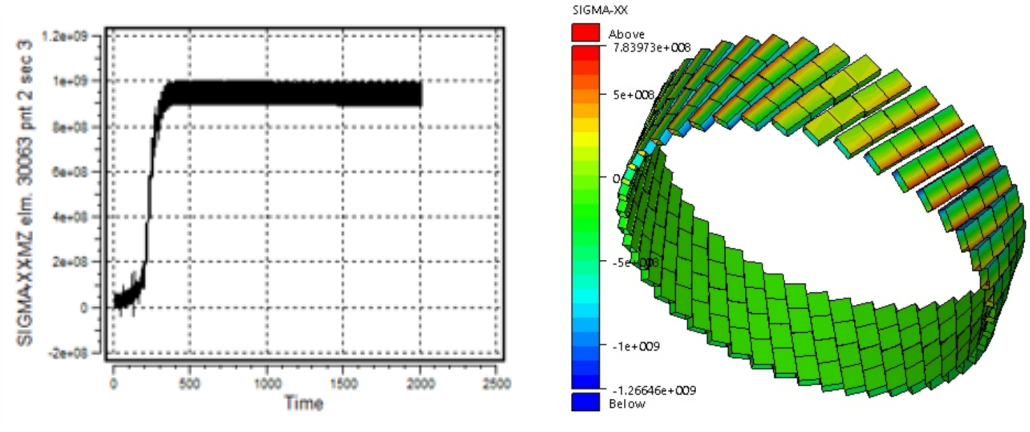


Figure 9.15: RUC Model for tensile armour lateral buckling

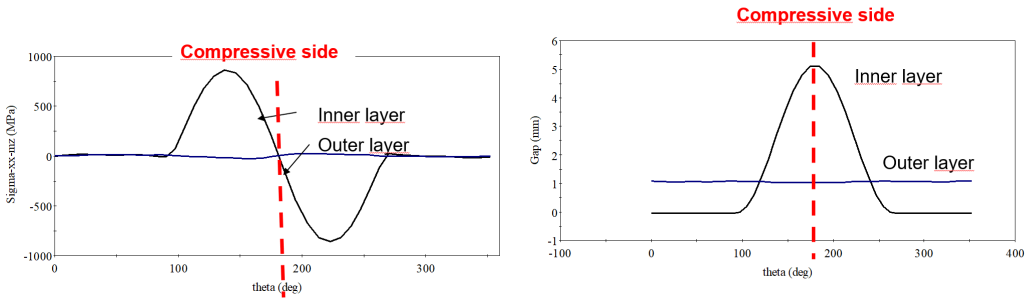
9.5.4 Results

The results are shown in Fig. 9.16 and Fig. 9.17. The maxium bending stress is stabilized quite rapidly due to the gap closure effect, quite in accordance with the observations of (Lukassen, 2019) and at a reasonable stress level, the latter being a result of the 0.93 fill factor. The computing effort in such models is small, which means that thousands of cycles can be efficiently processed.



(a) Bending stress versus time (b) Post buckling stress distribution

Figure 9.16: Development of stress during tensile armour buckling



(a) Bending stress distribution (b) Gap distribution

Figure 9.17: Stress and gap distributions around circumference

Table 9.5: FLEXIBLE PIPE DATA

Part	**Parameter**	**6" Riser**	**8" Riser**	**14" Jumper**
Inner tensile layer	Outer diam. (m)	0.201	0.276	0.442
	Pitch (m)	1.263	1.474	2.247
	Lay angle (deg)	26.2	30	31.5
	Wire size (mm)	310	512.5	415
	Num. of wires	52	54	70
Outer tensile layer	Outer diam. (m)	0.209	0.289	0.452
	Pitch length (m)	1.318	1.525	2.345
	Lay angle	-26.2	-30.3	-31.0
	Wire size (mm)	3ÃŮ10	5ÃŮ12.5	4ÃŮ15
	Num. of wires	54	56	72
Steel properties	Yield str. (MPa)	650	1350	1350
	E-mod. (GPa)	210	210	210
	Poisson's ratio	0.3	0.3	0.3
Anti bird-caging tape	Outer diam. (m)	0.212	0.292	0.455
	Pitch (m)	0.075	0.025	0.140
	Lay angle (deg)	83.5	88.4	-84.4
	Tape size (mm)	160	1.81.3	160
	Num. of windings	1	8	2
	Elastic modulus (GPa)	27	27	27
Outer sheath	Outer diam. (m)	0.225	0.314	0.477
	Thickness (mm)	6.0	10.0	10.0
	Elastic modulus (MPa)	400	400	400
Num. of inner layer pitches		3.96	3.39	3.34

Table 9.6: TEST CONDITIONS

Case	Applied compression (kN)	Bending radius (m)
6.1	265	11
6.2	80	11
6.3	210	11
6.4	160	11
6.5	265	8
8.1	700	12
8.2	300	12
8.3	400	12
14.1	277	21
14.2	269	8
14.3	411	9
14.4	950	12

Table 9.7: NUMERICAL RESULTS VERSUS TEST RESULTS IN TERMS OF FAILURE/NO FAILURE

	Experiment			Numerical	
Case	Cycles	Rotation (deg.)	Failure	Rotation (deg.)	Failure
6.1	204	45 (increasing)	Failure	10 (increasing)	Failure
6.2	800	< 1	No failure	3 (slowly increasing)	No failure
6.3	392	45 (increasing)	Failure	11 (increasing)	Failure
6.4	1200	3 (slowly increasing)	No failure	3 (increasing)	Failure
6.5	151	45 (slowly increasing)	Failure	10 (increasing)	Failure
8.1	1200	27 (slowly increasing)	Failure	21 (increasing)	Failure
8.2	1200	< 1	No failure	3 (slowly increasing)	No failure
8.3	1200	15 (slowly increasing)	No failure	18 (increasing)	Failure
14.1	6000	< 1	No failure	5 (slowly increasing)	No failure
14.2	1200	6.5 (increasing)	No failure	17 (increasing)	Failure
14.3	1200	27	Failure	27 (rapidly increasing)	Failure
14.4	1200	10 (rapidly increasing)	Failure	11 (rapidly increasing)	Failure

Table 9.8: FLEXIBLE PIPE DATA

Part	**Parameter**	**Value**
Inner tensile layer	Outer diam. (m)	0.2205
	Pitch (m)	1.375
	Lay angle (deg)	26.1
	Wire size (mm)	512.5
	Num. of wires	45
Outer tensile layer	Outer diam. (m)	0.2303
	Pitch length (m)	1.436
	Lay angle	-26.3
	Wire size (mm)	512.5
	Num. of wires	47
Steel properties	E-mod. (GPa)	210
	Poisson's ratio	0.3
Anti bird-caging tape	Outer diam. (m)	0.2353
	Tape thickness (mm)	2.4
	Elastic modulus (GPa)	26.4
Outer sheath	Outer diam. (m)	0.2563
	Thickness (mm)	10.5
	Elastic modulus (MPa)	1200

10 Searching for Errors

Below are a few tips that may be useful when you get an error return, and the reason is not clear to you.

- You have got a error message when a input card is read, but you can not find any errors in the card:
  Check the spelling of the next card identifyer. BFLEX2010 has a list of all the card keywords. If the spelling is incorrect BFLEX2010 does not recognice it, and assume that all belongs to the previous card.

- You have just made a long list of nodes and/or element connectivities with the **NOCOOR** or **ELCON** card, and BFLEX2010 goes in error:
  Each card has a maximum number of parameters that can be read. (Maximum is usually 1000) Use several cards to define the nodes or element connectivities.

11 Changelists

11.1 Changes in version 3.3.0

General changes:

- The functionality from the USAP software has been integrated into the BFLEX2010 framework. This includes new element types for the purpose of modelling helix components. These are the **HELIX231**, **HELIX233**, **HELIX234** and **HELIX235**. In addition, the **HELSPR437** element is included. This is used to link a pipe core system to a helix system of the above mentioned helices, and alov for relative sliding between the two systems.
- The USAP integration also accomodate for the **AUTOSTART** option, see **CONTROL**.
- Default selection of friction model has changed from 0 to 1 (old to new) when **MATERIAL** .. **SHEARMODEL** is applied.
- Secoond order terms of Green strain have been implemented for the helical elements. This implies that the behaviour in twist is better captured. The lay angle can in principle shift sign, still under the small strain assumption.

Input changes:

ELCON - CHANGED - Additional element types allowed: **HELIX231**, **HELIX233**, **HELIX234**, **HELIX235** and **HELPR437**

ELPROP - CHANGED - Additional element types allowed: **HELIX**, **SZHELIX** and **HELSPRING**

CONTROL - CHANGED - Additional initial configuration **AUTOSTART** allowed for.

ELPROP SHEARMODEL - CHANGED - Default friction model changed from old to new

DYNPLOT - CHANGED - Optional parameter for several plots to

ELHIST - NEW - Modifying element properties for selected elements according to a specified history

11.2 Changes in version 3.2.0

General changes:

- Introduction of the new element type HSHEAR364 which is a thick shell version of HSHEAR363. This means that the trough thickness radial stress can be described. This requires one radial node to be defined on each side in addition to the beam centre nodes, see Section 2.13. Note that it is not allowed to combine HSHEAR363 with HSHEAR364 when defining the flexible pipe cross-section by applying 353FLEXCROSS, see Section 2.10.2.

- The application of HSHEAR364 enables the user to describe contact between concentric layers by means of the new contact element HCONT464. This procedure then basically replaces the CAFLEX algorithm inherent in the ITCODE options contained in FLEXCROSS, see Section 2.10.1

- Manual modelling by application of HSHEAR364 by means of the element property options **SHEARHELIX** and **SHEAR2HELIX**, see Section 2.20.12 and Section 2.20.13 in combination with HCONT464 with **CONTINT**, see Section 2.5 and **LAYERCONTACT**, see Section 2.20.11 gives the possibility to include the friction hysteresis for any combination of concentric layers by means of manipulating the friction coefficient in the **FRICONTACT** material associated to HCONT464, see Section 2.26.12 and 2.20.

- Introduction of virtual integration points in combination with dynpostb, enables the user to apply BFLEX2010 as a filter to directly posprocess curvature and tension into time series of stress, see Section 2.12.

- Bugg with regard to the orientation of PFLEX normal curvature stresses when transferred to BOUNDARY corrected

- Test examples are included in the user manual including a comprehensive discussion on the BFLEX2010 development history and the features that are now included, see Section 1.8 .

- The DYNPOSTB utility program has been added, which can be used to postprocess time series results from the analysis that has been selected for storage by the new **DYNRES** input. See Sections 4 and 2.12 for details.

Input changes:

ELCON - NEW - HCONT364 Four noded thick shell element (two beam nodes plus inside and outside radial dofs), see Section 2.13.8.

ELCON - NEW - HCONT464 contact elements between concentric layers, see Section 2.13.8.

ELPROP - NEW - LAYERCONTACT layer contact element properties for HCONT464, see Section 2.20.11.

DYNRES - NEW - Select result storage for specified nodal, element or intergration point results for all time steps. These results will be stored for all time steps, regardles of the frequency of result storage that is specified in the **VISRES** input. See Section 2.12.

11.3 Changes in version 3.1.1

General changes:

- Interlayer contact element HCONT454 to simulate contact between armor wires (HSHEAR353 elements), see Sections 2.13 and 2.20.

Input changes:

353FLEXCROSS - CHANGED - is affected by introducing HCONT454 see Section 2.10.2.

MATERIAL - CHANGED - FRICONTACT the contact is treated based on using a fixed penalty parameter for the HCONT454 elements, see Section 2.26.12.

ELCON - NEW - HCONT454 contact elements helices in the same layer with HSHEAR353 elements, see Section 2.13.8.

ELPROP - NEW - LAYERCONTACT layer contact element properties for HCONT453, HCONT454 and HCONT463, see Section 2.20.11.

11.4 Changes in version 3.1.0

General changes:

- Two new elements HSHEAR353 and HSHEAR363 are implemented to allow arbitrary slide of the tendons. Accordingly, two new contact elements HCONT453 and HCONT463 are implemented to handle layer interaction. These new elements can handle rupture in the anti-buckling tape and cross section ovalisation. The application of these elements have been demonstrated by comparing with experimental results with good agreement.

- New shear interaction model. Shear deformations is likely to occur in the anti-wear layers that may increase the slip curvature depending on the cross-section characteristic. An analytical formulation for tendon equilibrium considering shear deformations have been implemented in BFLEX2010 to allow user specified shear interaction stiffness parameter. The implementation has been compared with two 4 inch pipes during the FPS 2000 project showing that the use of the shear interaction stiffness can significantly affect the moment curvature relation.

- Add command line options to run the BFLEX2010, these options can be randomly mixed, see Section 1.7:

  1) *-s MAXSTEP* user specified scaling factor to increase maximum number of load steps, default value is 1 (corresponds to 5000 load steps).

  2) *-m MEMFAC* user specified scaling factor to increase memory to run the analysis, default value is 1.0.

  3) *-n prefixname* run the analysis immediately using input file *prefixname.2bif* from command line or within a script.

- More options to run the PFLEX, see Chapter 5:

  1) *IPROC*, optional, procedure parameter, default value=2.

  2) *MOMCTR*, optional, bending moment scaling factor, default value=1.

  Based on the calibration studies carried out, the following alternives are recommeded for future use of the PFLEX:

  - Either, use the default model (BFLEX2010 3.0.9 of Summer 2016) i.e. *IPROC=2*, in combination with *MOMCTR=0* for high tension cases, and *MOMCTR=1* for small tension cases.

  - or, use the new analytical model i.e. *IPROC=0*, in combination with *MOMCTR=0* for high tension cases, and *MOMCTR=1* for small tension cases.

- Add command line option to run PFLEX:

  Syntax: *pflex.exe -n prefix*

  -n: avoid interactive entering of *.raf* file name and controlling parameters. The program will take *prefix.pif* as the input file.

  Content of a typical *prefix.pif* file is:

```
NFILES
N
CHRAF       INODE ILOCAL [DELTA0 MOMCTR IPROC]
RAF_1       INODE_1 ILOCAL_1 DELTA_1  MOMCTR_1 IPROC_1
...
RAF_I       INODE_I ILOCAL_I DELTA_I  MOMCTR_I IPROC_I
...
RAF_N       INODE_N ILOCAL_N DELTA_N  MOMCTR_N IPROC_N
```

  where

  N Number of analysis.

  CHRAF The prefix of the 2bif file.

  INODE Node position for the PFLEX model.

  ILOCAL Local deformation parameter.

  DELTA0 Initial ovalization, *optional*.

  MOMCTR Bending moment scaling parameter, *optional*.

  IPROC Procedure parameter, *optional*.

- Add command line option to run BOUNDARY:

  Syntax: *boundary.exe -n prefix*

  -n: avoid interactive entering of *.raf* file name and controlling parameters. The program will take *prefix.boi* as the input file.

  Content of a typical *prefix.boi* file is:

```
NFILES
N
CHRAF INODE FIMOD ILOCAL ISTRES NONLIN ISTPFR ISTPBE ICODE TRCURP
RAF_1    INODE_1FIMOD_1ILOCAL_1ISTRES_1NONLIN_1ISTPFR_1ISTPBE_1ICODE_1TRCURP_1
...
RAF_I    INODE_IFIMOD_IILOCAL_IISTRES_INONLIN_IISTPFR_IISTPBE_IICODE_ITRCURP_I
...
RAF_N    INODE_NFIMOD_NILOCAL_NISTRES_NNONLIN_NISTPFR_NISTPBE_NICODE_NTRCURP_N
```

The meaning of each parameter can be referred to Chapter 6.

- Add command line option to run LIFETIME:

  Syntax: *lifetime.exe -n prefix*

  -n: avoid interactive entering of *.lif* file name and controlling parameters. The program will take *prefix.lif* as the input file.

  Content of a typical *prefix.lif* file is:

```
CHOUT
PREFIX.LOF
NFILES
N
CHRAF       CHINCL NCYCLE NSTART NEND IFLAG
RAF_1       CHINCL_1NCYCLE_1NSTART_1NEND_1IFLAG_1
...
RAF_I       CHINCL_INCYCLE_INSTART_INEND_IIFLAG_I
...
RAF_N       CHINCL_NNCYCLE_NNSTART_NNEND_NIFLAG_N
```

  where CHOUT is the result file name, full name needs to be specified. The meaning of other parameters can be referred to Chapter 7.

- Add command line option to run BFLEX2010POST:

  Syntax: *bflex2010post.exe -n prefix*

  -n: avoid interactive entering of *.2bpi* file name and controlling parameters. The program will take *prefix.2bpi* as the input file.

- Add command line option to run BPOST:

  Syntax: *bpost.exe -n prefix*

-n: avoid interactive entering of *.bpi* file name and controlling parameters. The program will take *prefix.bpi* as the input file.

Input changes:

CROSSECTION - NEW - 353FLEXCROSS is new option to model the flexible pipe using HSHEAR353 and HSHEAR363, see Section 2.10.2.

ELCON - NEW - HSHEAR353 is a 4 noded 26 dof curved beam element dedicated to the modelling of helices, see Section 2.13.7.

ELCON - NEW - HSHEAR363 is a 3 noded 15 dof beam-shell element dedicated to the modelling of the pressure armour, the anti-buckling tape and the plastic layers, see Section 2.13.7.

ELCON - NEW - HCONT453 is contact element to model contact between two helix layers of HSHEAR353 elements, see Section 2.13.8.

ELCON - NEW - HCONT463 is contact element to model contact between HSHEAR353 and HSHEAR363 elements, see Section 2.13.8.

ELPROP - NEW - SHEARHELIX Shear element properties for hshear353 and the tubular option for HSHEAR363, see Section 2.20.12.

ELPROP - NEW - SHEAR2HELIX Shear element properties for the helix option for HSHEAR363, see Section 2.20.13.

ELPROP - NEW - SHEARMODEL Shear interaction model selection, see Section 2.20.14.

MATERIAL - NEW - FRICONTACT the contact is treated based on using a fixed penalty parameter for the HCONT453 and HCONT463 elements, see Section 2.26.12.

11.5 Changes in version 3.0.9

General changes:

- External bug related to external pressure has been corrected. No change to the user manual and input file. It should be emphasized that the element number (not the pressure) should be in negative if external hydrostatic pressure is applied.
- Bug related to moment/stress related to itcode31 was detected and corrected. The initial slope of the shear interaction curve in ITCODE 31 is now purely based on analytical theory, whereas in 2.19.1 this was based on integrating a stick-slip model.
- Changes in PFLEX. The following actions have been taken to improve the above with respect to the PFLEX:

  1) To include the tension curvature term.

  2) To modify the support condition to represent hard contact.

  3) To write the maximum ovalization to the .plf file (measured at the outside of the pressure spiral and also to the screen.

  4) To automatically increase the meshing of the pflex model by a separate input parameter in BFLEX2010POST.

  5) To replace the displacement norm with force norm.

  6) Allow more iterations (set to 500)
- Changes in bell mouth modelling. The geometry of the bell mouth is now calculated based on analytical expressions instead of numerical integration. This removes numerical inaccuracies that could lead to inaccurate estimates of the contact between the riser and the bell mouth. The analytical expression is singular for zero change in curvature along the bell mouth (constant curvature bell mouth). Thus the curvature is taken as constant if the maximum curvature differs by the initial curvature by a fraction of less than 0.01.
- Changes in bell mouth contact spring. A transition zone in the stiffness in the normal direction is added to the CONT152 element. The transition zone extends 10^{-3} times the smallest penetration value given in the material card to both sides of zero penetration. The shape is of the transition zone is that of a half sine wave. The transition zone will in theory not lead to any changes in the equilibrium state, only enhance convergence.

  The contact spring was previously defined as:

EXAMPLE:

```
#        name  type     eps    sig
MATERIAL bellz hycurve  -1000  -1e5
                         1000   1e5
```

The contact spring is now recommended to be defined as:

EXAMPLE:

```
#        name  type     eps    sig
MATERIAL bellz hycurve  -1000  -1e5
                        -10    -1e3
                         1000   1e5
```

Bibliography

Berge, S., Engseth, A., Fylling, I., Larsen, C., Leira, B., Nygaard, I., and Olufsen, A. (1992). Handbook on design and operation of flexible pipes. Technical report, SINTEF - Structures and Concrete.

Boru, M. E. (2021). Viv fatigue of dynamic power cables applied in offshore wind turbines. Master thesis, NTNU, Trondheim, Norway.

Dai, T., Sævik, S., and Ye, N. (2017). Friction models for evaluating dynamic stresses in non-bonded flexible risers. *Marine Structures*, 55:137 – 161.

Dai, T., Sævik, S., and Ye, N. (2018). An anisotropic friction model in non-bonded flexible risers. *Marine Structures*, 59:423 – 443.

Dai, T., Sævik, S., and Ye, N. (2020). Experimental and numerical studies on dynamic stress and curvature in steel tube umbilicals. *Marine Structures*, 72:102724.

Feret, J. and Momplot, G. (1989). *CAFLEX, Computer program for capacity analysis of flexible pipes, Theory Manual*. SINTEF and IFP, Trondheim, Norway.

Longva, V. and Sævik, S. (2013). A penalty-based contact element for pipe and 3d rigid body interaction. *Engineering Structures*, 56:1580 – 1592.

Lukassen, T. V. (2019). *Constitutive Behavior of Tensile Armor Wires in Unbonded Flexible Pipes*. PhD thesis, Technical University of Denmark, Lyngby, Denmark.

Lukassen, T. V., Gunnarsson, E., Krenk, S., GlejbÃÿl, K., Lyckegaard, A., and Berggreen, C. (2019). Tension-bending analysis of flexible pipe by a repeated unit cell finite element model. *Marine Structures*, 64:401–420.

Lutchansky, M. (August 1969). Axial stresses in armour wires of bent submarine cables. *ASME Journal of Engineering for Industry*, 91:687–693.

Østergaard, N. (2012). *On Lateral Buckling of Armouring Wires in Flexible Pipes*. PhD thesis, Aalborg University, Aalborg, Denmark. ISBN 87-91464-35-8.

Sævik, S. (1992). *On Stresses and Fatigue in Flexible Pipes*. PhD thesis, NTH, Trondheim, Norway.

Sævik, S. (1999). A finite element model for predicting longitudinal stresses in non-bonded pipe pressure armours. In *Proc. of MARINFLEX*, London, UK. ASME.

Sævik, S. (2005-02-14a). USAP - *Test Manual, Marintek report no. 700254.00.03*. MARINTEK, Trondheim, Norway.

Sævik, S. (2005-02-14b). USAP - *Theory Manual, Marintek report no. 700254.00.01*. MARINTEK, Trondheim, Norway.

Sævik, S. (2011). Theoretical and experimental studies of stresses in flexible pipes. *Computer and Structures*, 89:2273–2291.

Sævik, S. (2022). BFLEX2010 - *Theory Manual*. MARINTEK, Trondheim, Norway.

Sævik, S., Giertsen, E., and Olsen, G. P. (1998). A new method for calculating stresses in flexible pipe tensile armours. In *Proc. of OMAE'1998*.

Sævik, S., Gray, L., and Phan, A. (2001). A method for calculating residual and transverse stress effects in flexible pipe pressure spirals. In *Proc. of OMAE'2001*.

Sævik, S. and Igland, R. T. (2002). Calibration of a flexible pipe armour stress model. In *Proc. of OMAE'2002*, Oslo, Norway. ASME. OMAE 2002-28092.

Sævik, S. and Li, H. (2013). Shear interaction and transverse buckling of tensile armours in flexible pipes. In *Proc. of OMAE'2013*.

Skallerud, B. (1991). Structural damping in a wellsttream pipe. Final report SINTEF Report STF71 F91059, SINTEF.

Skeie, G., Sødahl, N., and Steinkjer, O. (2012). Efficient fatigue analysis of helix elemenets in umbilicals and flexible risers: Theory and applications. *J. of Applied Mathematics*, (246812).

Zhou, C., Sævik, S., Ye, N., and Ji, G. (2015). Effect of lay angle of anti-buckling tape on lateral buckling behavior of tensile armors. In *Proc. of OMAE'2015*.

Index

Symbols

Boundary 11
Lifetime 11
Pflex 11
Bflex2010post 11
Bpost 11
Bflex2010 11
353FLEXCROSS 57

A

ASCII 22
AUTOSTART 39

B

BEAM 80, 82
BENDSTIFF 61
BENDSTIFF-BEND 61
BFLEX 48
BFLEX2010
– Input data 171
BFLEX2010POST 170
– BFPOST 183
– ELPLOT 173
– ENPLOT 183
– FAPLOT 180
– General 170
– GLPLOT 177
– GNPLOT 176
– GRPLOT 184
– IPPLOT 172
– NOPLOT 171
– NRPLOT 180
– RSPLOT 182
BFPOST 183
BODY 79, 100
BONCON 22, 27
BOUNDARY 192
BOX 45
BPOST 196
– ELPLOT 200
– General 196
– GLPLOT 203
– GNPLOT 201
– Input data 196
– NOPLO2 198
– NOPLOT 196

C

CABLE 75, 101
CLOAD 22, 28
COMPIPE 75, 98
CONEQ 29, 31
CONSTR 22, 29
CONT 75
CONTACT 80, 135
CONTINT 22, 34
CONTROL 22, 37
COSUPR 22, 41
COSURFPR 22
COSURFR 42
CROSSECTION 22, 52
– 353FLEXCROSS 57
– BENDSTIFF 61
– BENDSTIFF-BEND 61
– FLEXCROSS 53
– NLBENDSTIFF 62
– NLBENDSTIFF-BEND 62
CROSSGEOM 22, 44
– BFLEX 48
– BOX 45
– GENERAL 46
– PIPE 45

D

DYNCONT 22, 67
DYNRES_ 22, 68
DYNRES_E 69
DYNRES_I 70
DYNRES_N 68

E

EIGEN 38
ELASTIC 127
ELASTOPLASTIC 127
ELCON 22, 71
ELDAMP 22, 79
ELECC 22, 82
– BEAM 82
– RADIUS 85
– STINGER 83
ELHIST 88
ELLOAD 22, 90
ELMASS 22, 91
– BEAM 80
– CONTACT 80
– RALEIGH 81
ELORIENT 22, 92
ELPLOT 173, 200
ELPROP 22, 95
ENPLOT 183
ENVRES 117
ENVRES_E 23, 118
ENVRES_I 23, 118
ENVRES_N 23, 117
EPCURVE 129

F

FAPLOT 180
Fatigue data 63
FATPROP 23, 119
FLEXCROSS 53
FRICONTACT 140
FRICVISC 142

G

GENERAL 46
GENSPRING 105, 149
GEOM 23, 121
GLOBAL 30, 32
GLPLOT 177, 203
GNPLOT 176, 201
GRPLOT 184

H

HCONT 78
HEAD 23
HEADING 123
HELIX 76
HELSPR 78
HELSPRING 150
HISTCONEQ 29, 33
HSHEAR 77
HYCURVE 131
HYPERELASTIC 148

I

INISTR 23, 123
IPPLOT 172
ISOCONTACT 139
ISOHYCURVE 132
ISOKXYCONTACT 140

L

LAYERCONTACT 109
LIFETIME 194
LINEAR 126
LOCAL 30, 31
LOCSL 32

M

MATERIAL 23, 124

– CONTACT 135
– ELASTIC 127
– ELASTOPLASTIC 127
– EPCURVE 129
– FRICONTACT 140
– FRICVISC 142
– GENSPRING 149
– HELSPRING 150
– HYCURVE 131
– HYPERELASTIC 148
– ISOCONTACT 139
– ISOHYCURVE 132
– ISOKXYCONTACT 140
– LINEAR 126
– R_CONTACT 143
– RESULTANT 133
– SEA 133
MOVE_GROUP 23
move_group 150

N

NLBENDSTIFF 62
NLBENDSTIFF-BEND 62
NOCOOR 23, 150
NODPROP 23, 153
NOORIENT 23, 155
NOPLO2 198
NOPLOT 171, 196
NRPLOT 180

P

PDISP 29
PELOAD 23, 156
PFLEX 189
PILOAD 23, 157
PIPE 45, 74, 96

R

R_CONTACT 143
RADIUS 85
RALEIGH 81
REEL 23, 159
RESTART 38
RESULTANT 133
ROLLER 102
RSPLOT 182

S

SEA 78, 133
SHEAR2HELIX 111
SHEARHELIX 110
SHEARMODEL 112
SIMPOST
– FAPLOT 180
– RSPLOT 182
SPRING 76
STINGER 83
STRESSFREE 38

T

TABLE 161
THIST 23, 163
TIMECO 23, 165
TLOAD 23, 166

V

VISRES 23, 167