RISER PRODUCTION LINE

BS-engine User Manual

Ulta deep LLC

Version No.: 1.0.0., Rev. 0
Document No.: N/A
Date: 06-04-2016

Project name:	Riser Production Line
Report title:	BS-engine User Manual
Customer:	Ulta deep LLC
Contact person:	Knut Ivar Ekeberg
Date of issue:	06-04-2016
Project No.:	N/A
Organisation unit:	Riser Technology
Version No.:	1.0.0., Rev. 0
Document No.:	N/A

DNV GL DNV GL Oil & Gas
BDL Riser & Umbilical
P.O.Box 300
1322 Høvik
Norway
Tel: +47 67 57 99 00
NO 945 748 931 MVA

Task and objective:
User manual for the software BS-engine.

Prepared by:	Verified by:	Approved by:
Nils Sødahl Vice President	Geir Skeie Senior Principlal Specialist	Oddrun Steinkjer Head of Section

☐ Unrestricted distribution (internal and external)
☒ Unrestricted distribution within DNV GL
☐ Limited distribution within DNV GL after 3 years
☐ No distribution (confidential)
☐ Secret

Keywords:
Non-linear FE analysis, Bend stiffener, non-linear material

Reference to part of this report which may lead to misinterpretation is not permissible.

Rev. No.	Date	Reason for Issue	Prepared by	Verified by	Approved by
0	06-04-2016	First issue	NIS	GES	OSTEI

Table of contents

1 INTRODUCTION

1.1 General description

BS-engine is tailor made software for static analysis of bend stiffeners (BS) by means of finite element (FE) analysis.

The FE solver is a self-contained subroutine package with a well-defined data structure interface to the driver program (i.e. main program, in this context the BS-engine). The driver program is typically tailor-made for each application to facilitate easy user interface and analysis set-up requirements (e.g. structural modelling and meshing, set-up of data structure for FE solver, load application sequence etc). The following functionality of the FE solver is of relevance for BS modelling/analysis:

- 3D non-linear static solver with an iterative load incrementation solution scheme.
- A co-rotational formulation is applied allowing for ‘unlimited’ displacements and rotations in 3-space
- 3D beam elements with linear material modelling and geometric stiffness
- 3D beam elements with non-linear elastic material modelling in bending (moment vs curvature) and geometric stiffness
- Boundary conditions (fixed/free) at any node in the FE model
- Application of point loads at any node in the FE model
- Loading due to volume forces (weight and buoyancy)

The user input to BS-engine is described in this document.

1.2 Running the program

The following command line options are available.

Option	**Action**
prog	Interactive execution of program *prog*
prog –b	program *prog* is executed in batch mode with input read from default input file prog.inp
prog –b filename	program *prog* is executed in batch mode with input read from input file *filename*
prog –v	Program version information, program description and revision history is written to standard output

In this context *‘prog’* means BS-engine

2 INPUT SPECIFICATION

2.1 General

The main input specification to is given in terms of a text input file (ASCII)

Input data are organised in terms of several data groups with a specific number of input data to be given for each data group. Each data group is identified by a *Data Group Identifier*. The following general rules apply:

1. Text lines starting with ** or ' are treated as comment lines.
2. The data groups may be given in arbitrary sequence on the input file
3. Data group identifiers are case insensitive

It is distinguished between 3 input types:

A – ASCII (text) input variable

I - Integer number

R – Real number

A basic input file example is shown in Appendix A.

2.2 Structural model specification

2.2.1 Modelling principles and assumptions

The structural model consist of the bend-stiffener itself as well as the riser. The following modelling assumptions apply:

- The structural model includes the bend stiffener body and the riser. The root end of the bend stiffener is fixed (all dof's), all other dof's in the model is free.
- A horizontal stress-free configuration is used, directed in positive x-axis, with bend stiffener root end at origin.
- A linear material modelling is applied for the riser.
- A linear or non-linear material model (or combinations) can be applied to model the bend stiffener properties
- It is assumed that the bend stiffener and riser will have the same curvature in bending. This means that the resulting bending properties of the combined model is found by adding the bending stiffness of the bend stiffener and the riser (i.e. possible gap or play between the bend stiffener and riser is not accounted for in the model)
- No shear load transfer (due to friction) between the bend stiffener and riser is accounted for. This means that the tensile load is carried entirely by the riser.
- The riser model starts at the bend stiffener root end and extends beyond the bend stiffener tip. In this way the interaction between the riser and the bend stiffener body is captured (possible local curvature maximum in the riser outside the bend stiffeners body is captured).

- The riser part beyond the riser tip should be sufficient long to disregard shear force and bending in the riser itself as boundary conditions (i.e. applied loads). This simplifies the load modelling significantly.

These assumptions allow for establishment of an efficient and robust FE solution scheme capturing the main physical effects, e.g :

- Linear/non-linear bending stiffness properties of the bend stiffener body
- Detailed modelling of the bend stiffener body geometry
- Geometric stiffness governed by the effective tension in the riser
- Interaction between bend stiffener body and riser

2.2.2 Bend stiffener specification

The bend stiffener structure is specified in terms of the following data:

- Inner diameter
- Outer diameter specified as a piece-wise linear function of length coordinate (from root end)
- Material density for each linear segment (for gravity load calculation, allowing for different materials to model e.g. possible joints)
- Each line segment can be specified as linear material (E-module) or non-linear (stress/strain curve)
- FE mesh will be generated to match the BS geometry using constant mesh within each linear segment)

The bend stiffener geometry is specified giving the following data cards:

Data group identifier:	**BEND STIFFENER DATA**

The following data lines must be given:

ID NSEG		
Parameter:	*Description:*	*Type*
ID	Inner diameter of the bend stiffener	R
NSEG	Number of segments for description of bend stiffener geometry	I

The following data line must be given NSEG times to describe the bend stiffener geometry:

LENGTH NSEL OD1 OD2 DENSITY ELTYPE MATERIAL		
Parameter:	*Description:*	*Type*
LENGTH	Segment length	R

NSEL	Number of elements. A constant element mesh is used within each segment	I
OD1	Outer diameter at end 1 (from root end)	R
OD2	Outer diameter at end 2	R
DENSITY	Material density	R
ELTYPE	Element type for FE analysis : LIN : Linear material modelling NOLIN or NOLIN-CYLINDER : Nonlinear material using cylindric element geometry. For this modelling alternative, the element mid-point is used for calculation of the non-linear moment/curvature of the cross section. NOLIN-CONE : Nonlinear material using conic element geometry. For this modelling alternative, the element end points as well as mid-point are used for calculation of the non-linear moment/curvature of the cross section. The conic geometry is hence accounted for in calculation of the element stiffness properties.	A
MATERIAL	Depending on element type, different material specifications must be given: LIN : EMOD (R) - E-modulus of bend stiffener material NOLIN-CYLINDER and NOLIN-CONE : MATID (A) – Identifier to non-linear stress/strain curve for the bend stiffener material	R/A

This data group is mandatory

2.2.3 Riser specification

Riser data is specified giving the following data group

Data group identifier:	**RISER DATA**

The following data line must be given:

SRIS NREL EI EA GT MASS		
Parameter:	*Description:*	*Type*
SRIS	Length of riser (from bend stiffener tip to free end)	R
NREL	Number of elements. A constant element mesh is applied	R
EI	Bending stiffness	R

EA	Axial stiffness	R
GT	Torsional Stiffness	R
MASS	Mass per unit lenght	R

This data group is mandatory

For user preference, this data group may also be specified as **UMBILICAL DATA**

2.3 Non-linear material modelling

2.3.1 Modelling principles and assumptions

The strategy for modelling of non-linear material properties for the bend stiffener is as follows:

- The bend stiffener material properties are specified in terms of a non-linear stress-strain curve typically established by testing at relevant temperature
- The material curve is specified for positive strain values. Symmetric properties are assumed in compression (to avoid modelling of shift in neutral bending axis)
- Numerical integration over the cross-sectional area is performed prior to FE analysis to establish the moment- curvature relationship for each element
- The internal force vector and stiffness matrix (bending related terms) are established by a 3-point Lobatto integration scheme over the length of each element during FE analysis. The integration points for the Loatto quadrature are element ends and element mid-point.
- Conical element geometry can be accounted for by deriving the moment-curvature relation at each integration point (using the actual outer diameter at each integration point when deriving the moment-curvature relation)

2.3.2 Material specification

The non-linear material properties are specified in terms of a strain-stress curve. The following conditions apply:

- Only non-negative values are accepted
- First data point must always be (0.,0)
- A minimum of 2 data points must be given
- Data points must be given as strictly increasing numerical values

Non-linear material data is specified giving the following data group:

Data group identifier:	**MATERIAL DATA**

The following data lines must be given to specify the non-linear material properties of the bend stiffener body:

MATID		
Parameter:	*Description:*	*Type*
MATID	Material identifier Must provide a unique reference in case more than one material is specified	A

NMAT		
Parameter:	*Description:*	*Type*
NMAT	Number of strain-stress data used to specify the bend stiffener material NMAT ≥ 2	I

The following data line needs to be given NMAT times to specify the material strain-stress curve:

STRAIN STRESS		
Parameter:	*Description:*	*Type*
STRAIN	Material strain	R
STRESS	Material stress	R

2.3.3 Moment-curvature calculation

The resulting moment/curvature relation for the cross-section is found by integrating the strain-stress over the cross-sectional area assuming plane sections remain plane under bending. The integration need to be carried out for a range of curvatures to establish moment-curvature table used in the FE analysis.

The following data group needs to be given to specify the moment-curvature tabulation:

Data group identifier:	**CURVATURE RANGE**

The following data lines must be given to specify the none-linear material properties of the bend stiffener body:

CURMAX NCURV		
Parameter:	*Description:*	*Type*
CURMAX	Maximum curvature to for tabulation of moment-curvature. Moment-curvature will by this specification be tabulated at NCURV equidistant points in the interval 0 .≤ curvature .≤ CURMAX CURMAX should represent a realistic upper bound value for the expected maximum curvature response of the bend stiffener	R
NCURV	Number of points in moment-curvature table. NCURV ≥ 2 NCURV should be selected to represent the nonlinear strain-stress curve of the material accurately, typically NCURV ≥ NMAT	R

2.3.4 Export of material data

The computed moment-curvature may be written to file. The intention with this functionality is to enable export of material data from BS-engine for use in other software. This may be specified by giving the following data group:

Data group identifier:	**EXPORT MATERIAL DATA**

The following data line must be given:

IMEX		
Parameter:	*Description:*	*Type*
IMEX	Data export format IMEX = 1 Tabular form IMEX = 2 Riflex input format	I

2.4 Loads

2.4.1 Riser end force

A nodal force can be specified at the free end of the riser. The load is acting in the x-z plane and is specified by magnitude and direction (relative to stress-free configuration corresponding to the longitudinal direction of the root-end fixation)

Nodal load is specified by the following data group:

Data group identifier:	**FORCE**

The following data line must be given:

RELANG TENSION		
Parameter:	*Description:*	*Type*
RELANG	Force direction (deg) relative to stress-free configuration. (positive y-rotation angle, i.e. positive in clock-wise direction)	R
TENSION	Effective tension	R

2.4.2 Weight load

Loads due to self-weight of the bend stiffener and riser may be included by specification of the following data group:

Data group identifier:	**WEIGHT LOAD**

2.5 FE analysis parameters

2.5.1 Solution strategy

The following solution scheme is applied in the non-linear static FE analysis:

- The loads are applied in multiple load increments with equilibrium iteration at each incremental load step until the full load is applied.
- The incremental load application starts from the stress-free configuration with update of coordinates and rotation matrices at each iteration cycle of each incremental load step.
- A Newton-Raphson equilibrium iteration is applied at each incremental load step.
- An adaptive load incrementation scheme is applied. This means that the load increment is adjusted automatically to improve numerical stability as the load incremetation process is progressing.

2.5.2 Specification of FE solution parameters

The following data group must be given to specify FE analysis parameters:

Data group identifier:	**FE ANALYSIS PARAMETERS**

The following data line must be given to specify the Newton-Rapshon equilibrium iteration parameters:

TOLNOR MAXIT		
Parameter:	*Description:*	*Type*
TOLNOR	Value of displacement norm, used as convergence criterion for the Newton – Rapshon iteration. Default value : 1.0E-07	R
MAXIT	Maximum number of iterations Default value : 30	I

The following data line must be given to specify the adaptive load incrementation scheme. Note that increments are specified in terms of percentage of full load (full load corresponds to 100)

DSINC DSMIN DSMAX		
Parameter:	*Description:*	*Type*
DSINC	Basis increment (percentage of full load). This should be specified as 'best estimate' increment size based on experience. Default value : 1.	R
DSMIN	Minimum increment (percentage of full load). Default value : 0.1	R
DSMAX	Maximum increment (percentage of full load). Default value : 10.	R

The load application will start out with the basis increment DSINC. The load increment will be adjusted between the minimum value (DSMIN) and the maximum value (DSMAX) as the load incrementation is progressing to improve the numerical stability. The target of the present adaptive algorithm is to achieve 3-5 iterations per load increment which is considered to be a good compromise between computational efficiency and numerical stability.

Note : The suggested default values should only be considered as initial recommendations. Better recommendations may be given at a later stage as more experience is gained for this particular application. Feedback from users is encouraged to further improve the solution strategy.

2.6 Result data storage

2.6.1 Formatted file storage

Formatted (i.e. text file) data storage of main results from the FE analysis may be specified by giving the following data group:

Data group identifier:	**FORMATTED RESULT DATA FILE**

2.6.2 Unformatted data storage

Unformatted (i.e.binary sequential file) data storage of main results from the FE analysis may be specified by giving the following data group:

Data group identifier:	**UNFORMATTED RESULT DATA FILE**

2.7 Input file termination

The data given as input must be terminated by 'END' to specify that this is the end of the input specification.

This data group is mandatory

Data group identifier:	**END**

3 OUTPUT FILE DESCRIPTIONS

3.1 Overview

Output files	**Description**
gfile.log	Echo of input data, warnings and error messages, FE results, post-processing
gfile_FEA.log	Element data, warnings and error messages from FE solver, echo of numerical solution scheme
gfile-material-data.dat	Material data (moment-curvature)
gfile.dat	Result data file. Binary/text file depending on user specification

Where 'gfile' denotes the generic file name derived from the input file excluding the file extension '.inp'. E.g. if input file 'example.inp' is specified this means that 'gfile' is 'example'.

3.2 Result data file description

The user may specify result storage in sequential text/binary file format. The content is identical. A high level description of file content is given in the following.

1) Key results (one line)
2) Total number of elements (NEL) and number of elements in bend stiffener (NELBS) (one line)
3) Bend stiffener results (NELBS+1 lines)
4) Coordinates and curvature, complete model (NEL+1 lines)
5) Element results, complete model (NEL lines)
6) Element results, riser only (NEL lines)

The content per item is identical to the edited printout on the 'BS-engine.log' file under heading 'FE analysis results', please consult the 'BS-engine.log' file for detailed descriptions.

4 REFERENCES

/1/

/2/

APPENDIX A
Basic input file example

Etter hvert …

About DNV GL

Driven by our purpose of safeguarding life, property and the environment, DNV GL enables organizations to advance the safety and sustainability of their business. We provide classification and technical assurance along with software and independent expert advisory services to the maritime, oil and gas, and energy industries. We also provide certification services to customers across a wide range of industries. Operating in more than 100 countries, our 16,000 professionals are dedicated to helping our customers make the world safer, smarter and greener.