Strain Concentrations in Pipes due to Concrete Coating

Unrestricted

EP 2006-5161

April 2006

Unrestricted **EP 2006-5161**

Strain Concentrations in Pipes due to Concrete Coating

by

R. Peek (SIEP EPT-PER)

Sponsor:	Sakhalin Energy Investment Company Pearl GTL Project
Reviewed by:	S. Draaisma, SIEP EPT-PER
Approved by:	R. Peek, SIEP EPT-PER
Date of issue:	April 2006
Account code:	B-000354-700-10.10.016
ECCN number:	EAR 99

SHELL INTERNATIONAL EXPLORATION AND PRODUCTION B.V., RIJSWIJK

Further electronic copies can be obtained from the Global EP Library, Rijswijk

SUMMARY

Concrete weight coating on subsea pipelines is typically not continuous over the field joints. This can result in significant strain concentrations in the field joint area, especially when the pipe is subject to plastic deformations and strain-based design is used. The reason for this is the extra bending stiffness provided by the concrete. As a result imposed bending deformations (e.g. due to the pipe going over the stinger of the laybarge, or to accommodate thermal strains by lateral buckling) tend to concentrate in the rather short field joint.

To address this without exaggerating the stiffening effect of the concrete, it is necessary to account for slip between the concrete weight coating and the pipe. For this purpose a model developed by STATOIL is implemented. The advantage of this implementation over that in STATOIL's MomKap program is that it allows the effect of concrete coating to be included in any finite element analysis using beam-type elements for the pipe, including geometric as well as material nonlinearities. Thus, for instance, concrete coating can be included in a model for lateral buckling of a pipeline. However the current implementation is 2-dimensional. The pipe must remain on one plane.

The approach is based on using 4 degrees of freedom per node: in addition to 2 displacement components and one rotation the relative slip displacement between the and the pipe and the concrete is introduced as the 4th degree of freedom.

This report includes a description of the formulation including assumptions made and validation of the implementation against results that have been provided by STATOIL.

KEYWORDS

Strain concentrations, pipes, concrete coating, subsea pipelines, field joints, Statoil, MomKap

TABLE OF CONTENTS

LIST OF FIGURES

1. THEORY AND IMPLEMENTATION

A special finite element as been developed to account for the effect of discontinuous concrete coating on the behaviour of pipes in the non-linear range according to the methodology of Verley and Ness [1][2]. The general modular finite element program (npex) developed at the University of Michigan, was used as the framework within which to develop this element. This program already contains two-dimensional pipe elements, which can be used together with the new concrete elements to model the interaction between the pipe and the coating.

Essentially the formulation accounts for the stiffness of the concrete in compression as well as the steel in tension or compression. In tension the concrete is assumed to be cracked , so that the stress is zero. However due to axial slip of the concrete over the pipe, the strain in the concrete is not that same as that in the pipe.

Consider the pipe in the X-Z plane, where X and Z are fixed cartesian coordinates, with X being the axial coordinate and Z the transverse coordinate. The concrete and the steel can be envisioned as separate beam type structures. Both are treated using the moderate deflection beam theory, including shear deformations. The transverse Z component of displacement $w=w(X)$ and rotation $\theta=\theta(X)$ is assumed to be the same for both the concrete and the pipe. However the axial displacements $u=u(X)$ are different for the steel and the concrete. The X-component of the force which the concrete exerts on the beam, depends on the difference in the axial displacements,

$$\Delta u(X) = u_{pipe}(X) - u_{concrete}(X) \quad (1.1)$$

according to an elastic-perfectly plastic type relationship, based on the maximum shear strength τ_y of the tar, and a mobilisation displacement needed to develop this maximum shear strength typically taken Δu_y=2mm[i]. In addition to the axial force transfer according to this elastic perfectly plastic relationship, transverse forces (in the Z direction) and body couples are also transferred between the steel and the concrete, as needed to ensure equality of $w(X)$ and $\theta(X)$.

For the finite element formulation in the X-Z plane, four degrees of freedom per node are used: axial displacement u for the pipe, transverse displacement w, rotation θ, and axial slip Δu the concrete relative to the steel.

The concrete is assumed to resist axial strains in compression only according to a parabolic stress-strain relationship given by

$$f_c = f_c' \; \{ 1 - [(\varepsilon_c' - \varepsilon_c)/\varepsilon_c']^2 \} \quad (1.2)$$

in which

f_c = axial stress in the concrete (positive for compression)

f_c' = compressive strength of concrete (typically around 40MPa)

ε_c = axial strain in the concrete (positive for compression)

ε_c' = strain at which the compressive strength of the concrete is reached (typically 0.2% used)

---

[i] Since the slip is typically well into the yielded region, this parameter has little influence on the results. However a finite value must be chosen to prevent ill-conditioning of the calculations and/or dividing by zero.

The above relation applies until the compressive strength of the concrete is reached. Beyond that point the concrete tends to loose strength. However to avoid convergence difficulties it is assumed that the stress in the concrete remains constant, if it is deformed beyond a strain of ε_c'.[ii] Under extension, the stress in the concrete is taken to be zero, due to cracking of the concrete.

In the npex program the concrete coating is implemented as a separate element according to a unified modular structure for adding new elements in npex. The routines are included in Appendix B, and the interface between these routines and the npex program is described in Appendix A. The model constructed has 4 degrees of freedom per node. The nodes are connected by pipe elements, and also by concrete coating elements where the pipe is coated. A node is always required where the coating ends (typically around 300mm to each side of the girth weld). For the nodes within a not-coated part of the pipeline (e.g. in the middle of the field joint area), and other nodes that do not have any concrete coating element attached to them, the 4th degree of freedom needs to be eliminated (by setting the corresponding displacement to zero).

ii If the calculated maximum compressive strain in the concrete is larger than ε_c', this means that crushing of the concrete occurs. This is regarded as a situation to be avoided by proper design, rather than analysed. To analyse it, one would need to modify this model to include the drop in concrete resistance upon crushing.

2. VALIDATION OF NPEX IMPLEMENTATION

STATOIL (Erik Levold) kindly provided a verification example calculated with their MOMKAP software, which embodies the implementation of the methods of Ness and Verley [1] described above. The problem solved represents a long string of 12.2m-long pipe joints under uniform bending. The concrete continues up to 35cm from the field joint. Thus at each field joint there is a 70cm-long section of pipe that is not coated with concrete. Additional parameters of this validation problem are as follows:

Pipe outer diameter	$D = 1078$ mm
Pipe wall thickness	$t_{pipe} = 30.8$ mm
Tar thickness	$t_{tar} = 6$ mm
Concrete coating thickness	$t_{conc} = 45$ mm
Concrete compressive strength	$f_c' = 40$ MPa
Concrete strain at ultimate	$\varepsilon_c' = 0.2\%$
Tar Shear Strength	$\tau_y = 0.1$ MPa

In view of the symmetries involved in a long string of pipe under pure bending (i.e. no axial load or shear force throughout), it is sufficient to model a section of pipe from the middle of a field joint, to the middle of a pipe joint, as shown in Figure 2.1. The pipe only to applied bending moment with no axial or shear forces, so it does not matter which end is fixed. In this case fixing the pipe at the field joint was more convenient for post-processing of the results.

For the stress-strain relation of the steel some curve fitting was needed, because the MOMKAP software uses a different form of the stress-strain curve, that that used by the pipe element in npex. Specifically the MOMKAP results are based on a Tvergaard type relationship given by

$$\varepsilon = (\sigma_y/E)\ \{[(\sigma/\sigma_y)^n - 1]/n + 1\} \qquad (2.1)$$

with $E = 207$ GPa, $\sigma_y = 372.6$ MPa and $n = 16.26$. On the other hand the NPEX results are based on a Dafalias stress-strain curve [3]. By choosing the Dafalias material parameters as

Initial yield stress	$\sigma_0 = 372.6$ MPa
Bounding stress	$\sigma_b = 492$ MPa
Hardening modulus	$h = 20{,}000$ MPa
Slope of the Bounding line	$E_0^p = 1250$ MPa

good agreement between the stress strain curves of the two models was achieved, as shown in Figure 2.2.

The approximately matching Tvergaard and Dafalias stress-strain curves are chosen here for comparison and validation purposes only. In actual applications it is important to choose the stress-strain curve the best describes the actual behaviour of the material, because the degree of strain concentration can be quite sensitive to the shape of the stress strain curve. Generally at strains above yield a flat stress strain curve (e.g. an elastic-perfectly plastic curve to represent Lüder's banding) leads to higher strain concentrations. However the rounded stress strain curves used here involve incipient yielding at stresses below the API yield stress (defined as the stress at 0.5% strain). Therefore at low strains, such as those on the stinger for pipelay in shallower water, an elastic perfectly plastic stress-strain curve gives rise to lower strain concentrations than those reported here for the Tvergaard or Dafalias stress-strain curves.

The comparison of the results provided by STATOIL from MOMKAP with those from the newly developed element in NPEX are shown in Figure 2.3 to Figure 2.6. A more precise definition of the terminology used therein is as follows:

- The "nominal bending strain" is defined as the maximum strain calculated from the curvature using elementary beam theory. In all cases it is calculated for the outer surface of the pipe. (Thus the nominal bending strain is equal to the curvature multiplied by one half the outer diameter of the steel pipe.)
- The "global curvature" is the relative rotation over the whole model divided by the length of the whole model. This is also equal to the average curvature over the length of the model.
- The "global bending strain" or "global nominal bending strain" or "average bending strain" is half the outer diameter of the steel pipe multiplied by the global curvature.

It is clear from the comparisons in Figure 2.3 to Figure 2.6 that the agreement is as good as might be expected given slight differences in the stress strain curves, in the discretisation or slightly different quantities plotted (e.g. nominal bending strain vs. actual strain).

Figure 2.7 and Figure 2.8 are provided to further illustrate the behaviour of the concrete coated pipe, and verify that the solutions are reasonable.

In Figure 2.7 it is seen that the axial force in the concrete increases more or less linearly with the distance from the field joint. This is due the shear stress τ_y transmitted across the tar coating between the concrete and the steel, due to the tendency for the coating to slip axially towards the field joint. The axial force in the concrete is compressive, and it is balanced by an equal and opposite force in the steel pipe, so that the net axial force is zero, as required from equilibrium.

Since the concrete cannot carry any tension, it must rely on a compressive axial force in order to be able to carry any bending moment. It is seen from Figure 2.8 that this results in a bending moment in the concrete that approximately proportional to the axial force of Figure 2.7. The total bending moment (from steel + concrete) is constant over the length of the pipe, as it must be for pure bending, from equilibrium considerations.

It is also clear from Figure 2.8 that the bending moment carried by the concrete is a small fraction of the total. Nevertheless, on the flat portions of the moment-curvature relationships, this can give rise to a considerable variation on the curvatures along the length of the pipe.

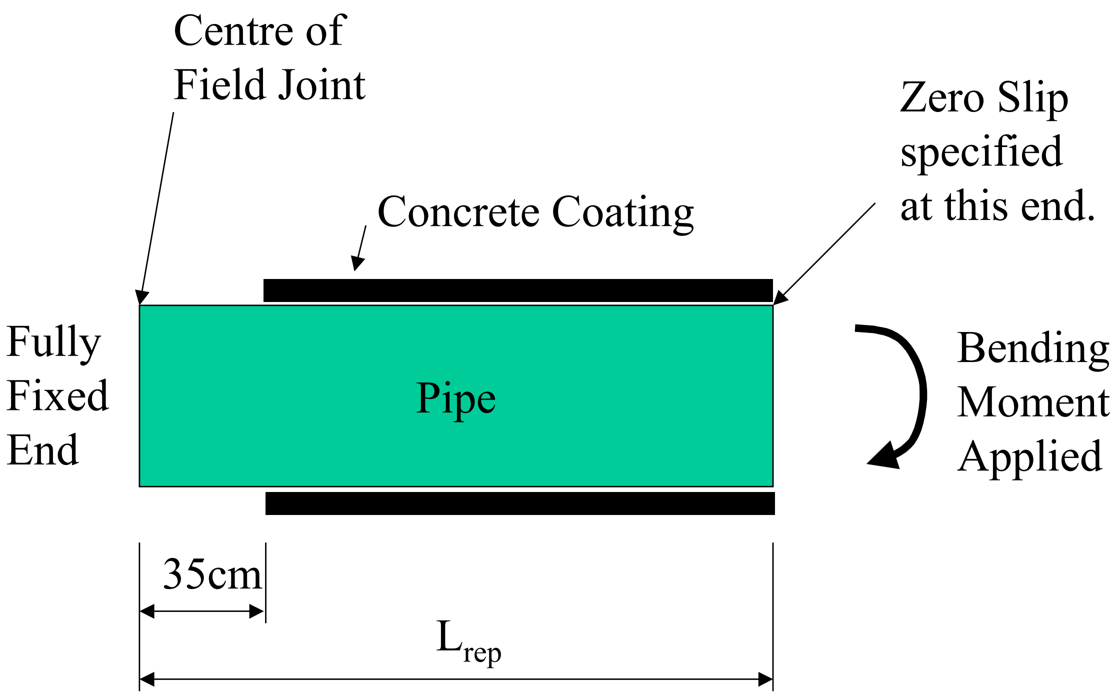


Figure 2.1: Finite element model used (L_{rep} is taken as 6.1m, i.e. half the joint length)

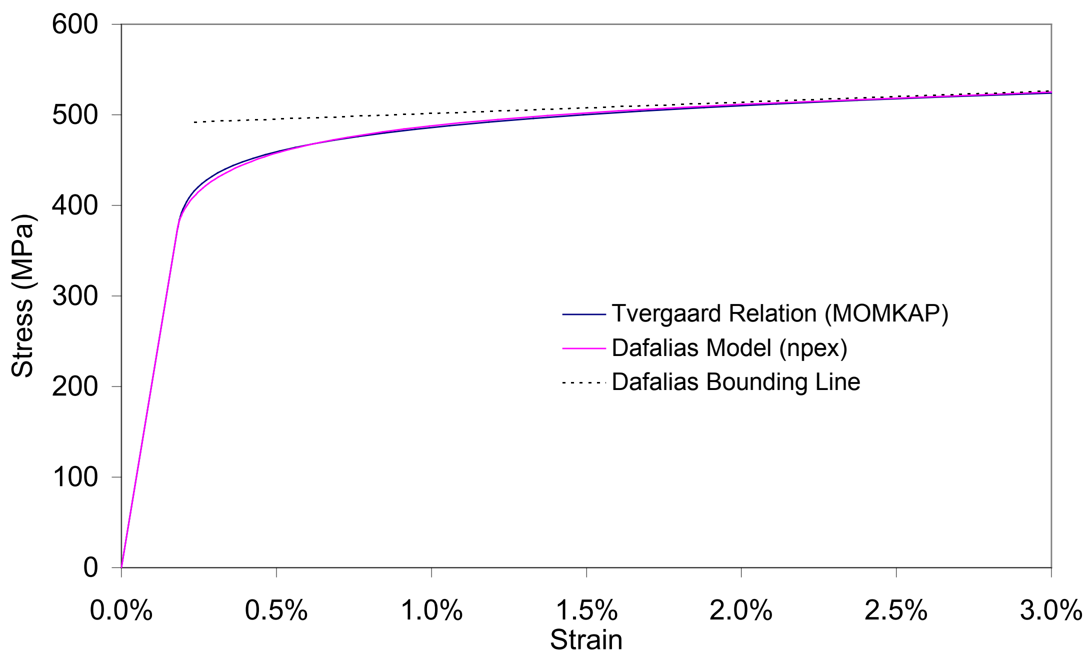


Figure 2.2: Comparison of the stress-strain curves used in the validation analyses.

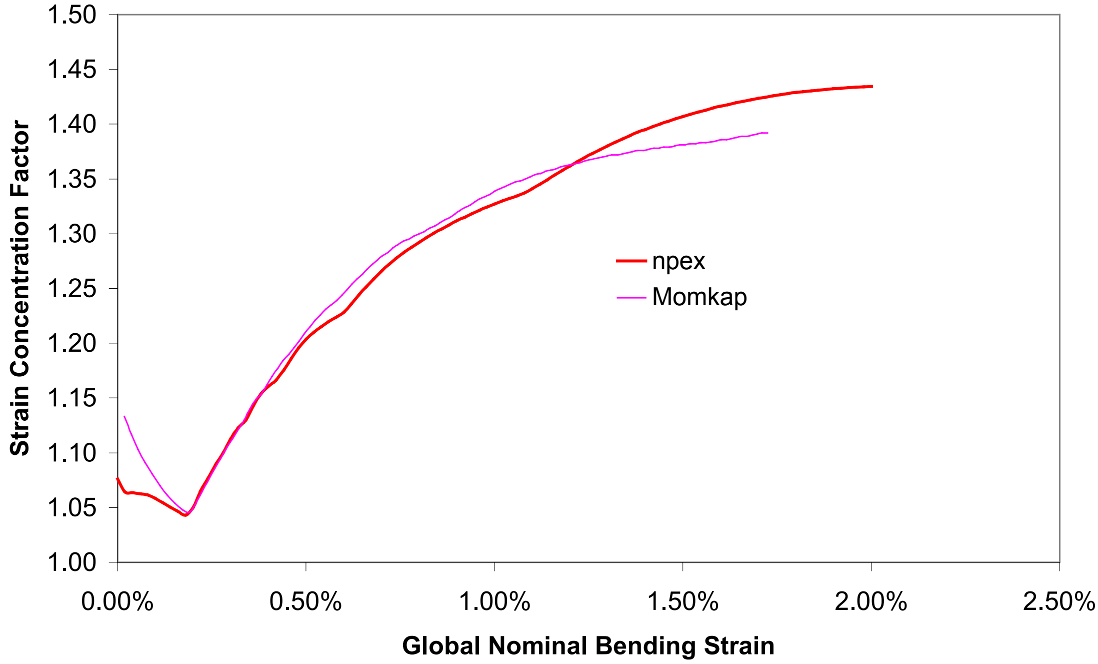


Figure 2.3: Comparison of Strain concentration factors (defined as the ratio of the curvature in the field joint, divided by the global average curvature).

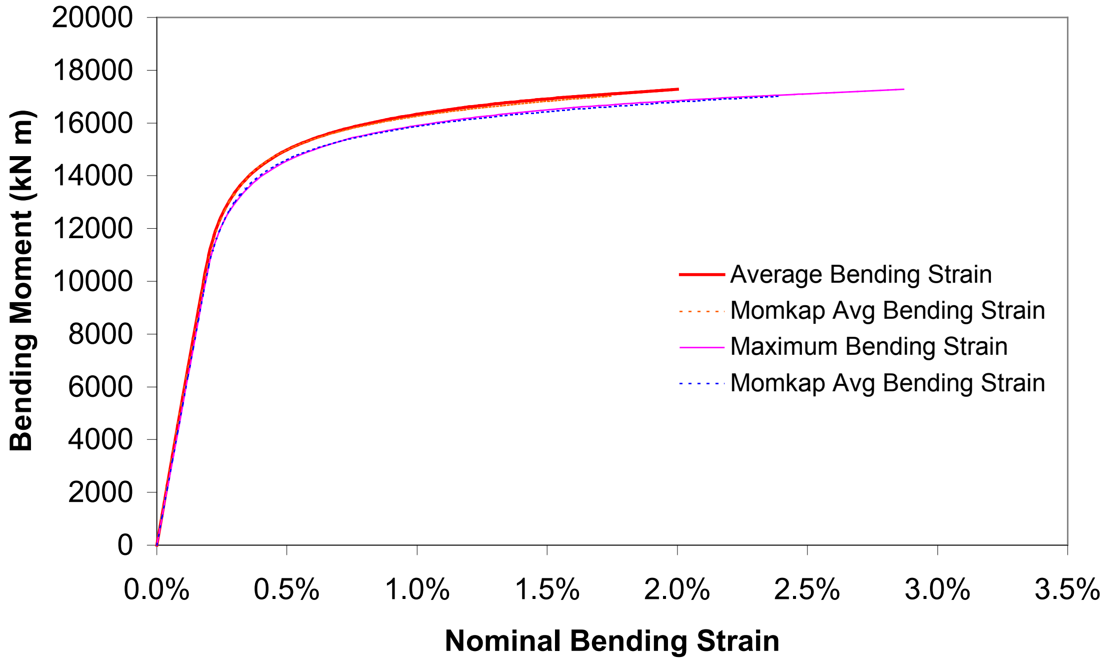


Figure 2.4: Comparison of moment curvature relationships.

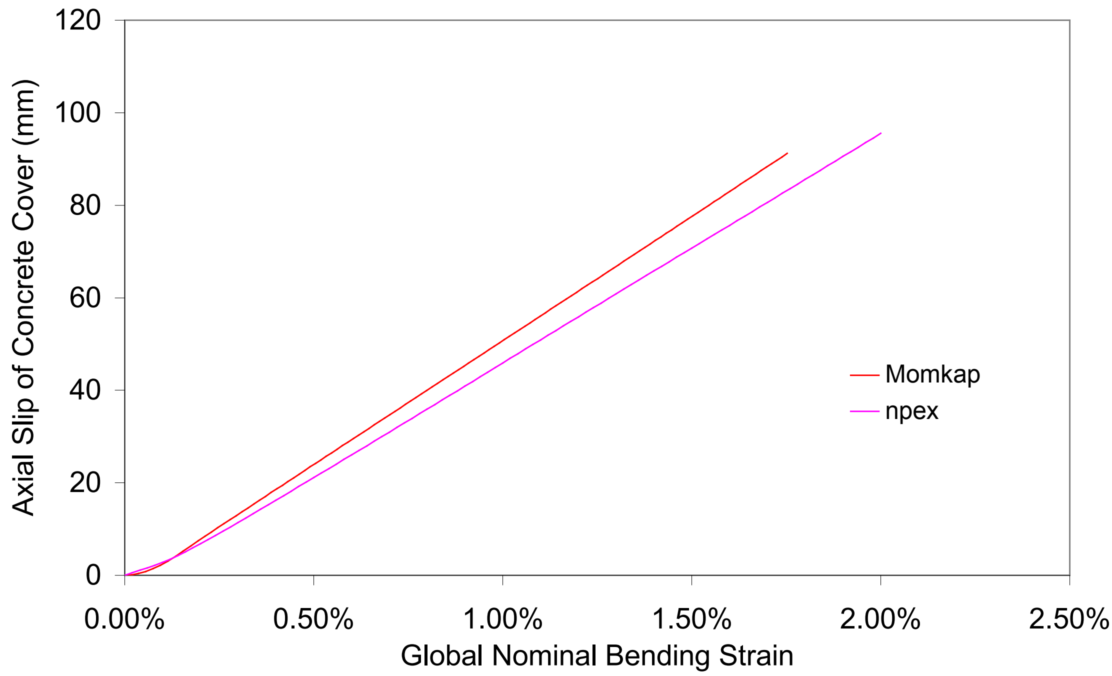


Figure 2.5: **Comparison of concrete axial slip at the field joint.**

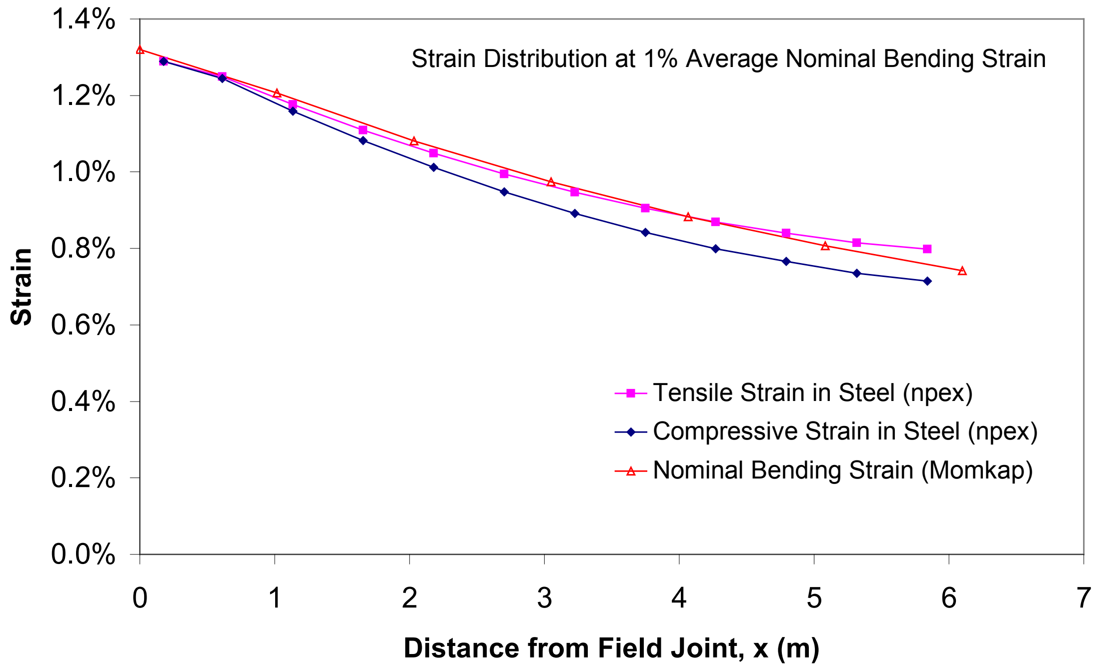


Figure 2.6: **Comparison of axial strain distributions along the length of the pipe, and a global nominal bending strain of 1%. (Note that npex results represent actual strains in the steel at the midsurface, whereas those from MOMKAP are nominal strains based on the curvature. The tensile strain in the steel is a bit higher than the compressive strain at the opposite side, some compression is carried by the concrete.)**

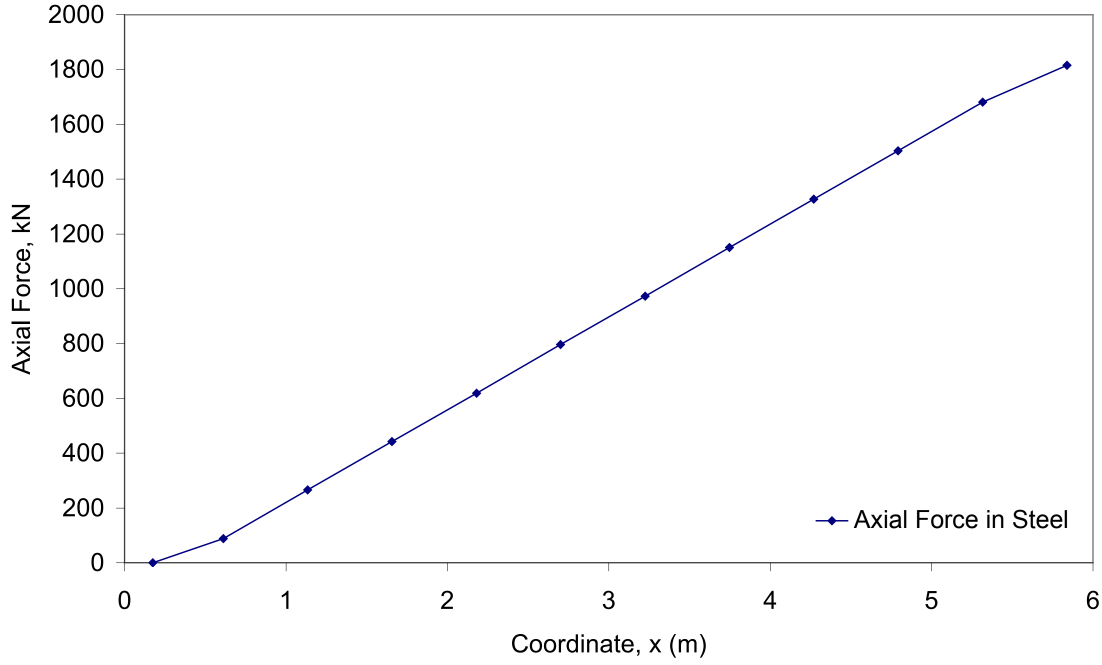


Figure 2.7: **Distribution of axial force (the compressive force in the concrete at any point is equal in magnitude to the tensile force in the steel). The linear variation in axial force arises because of a constant transfer of axial force per unit length from the steel to the concrete determined by the yield strength of the tar interface.**

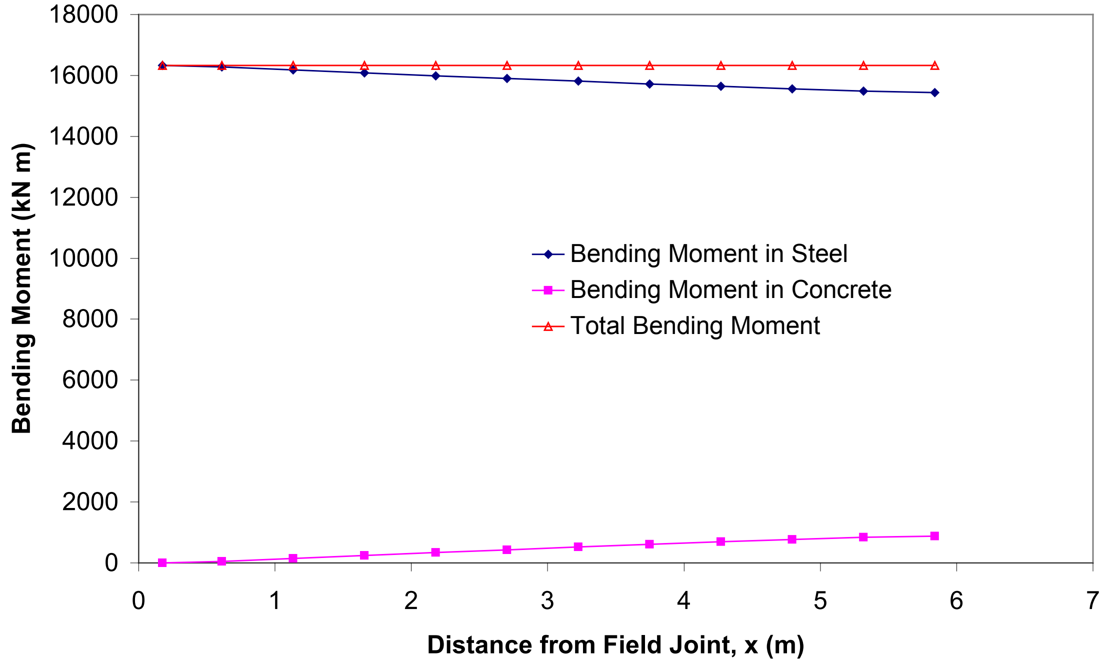


Figure 2.8: **Distribution of bending moments along the length of the half-joint from npex calculation. (Note that the total bending moment is exactly constant, as it must be from equilibrium under conditions of pure bending.)**

3. ESTIMATION OF THE EFFECTIVE CONCRETE-PIPE BOND SHEAR STRENGTH

Perhaps the most difficult model parameter to estimate is the effective shear strength of the bond between the concrete and the corrosion coating. Tests for this reported in [2] give the following shear capacities:

- 0.42 to 0.55MPa for asphalt enamel corrosion coating at room temperature
- 0.28 to 0.35MPa for Polyethene corrosion coating.

However complicating factors arise because the bond does not behave as an elastic perfectly plastic material, as is assumed in the analysis. These differ depending on the type of corrosion coating.

It must be borne in mind that the effective bond stress is not only a material property but also can compensate for assumptions made that are not an exact description of the real behaviour. One such assumption that the rotations of cross sections of the concrete are the same as those for the pipe, as illustrated in Figure 3.3a. I.e. it is assumed that the relative movement of the pipe and the concrete is purely axial, and is associated with equal axial slip all around the circumference. In reality the driving force for such axial slip comes from the compression side only. This tends to make cross sections of the concrete rotate with respect to the pipe, as shown in Figure 3.3b, so that more axial slip occurs on the compression side than on the tension side. The consequences of this behaviour are as follows:

a) At low strains there may not be enough axial slip on the tension side to develop the resistance τ_Y there. Thus the effective shear strength τ_Y is lowered, because the shear strength is not developed over the entire circumference.

b) At higher strains, the relative rotation can become so large that rings of the concrete tend to jam, thereby developing an extra frictional axial resistance. This can be enhanced by radial expansion of the pipe due to internal pressure. Of course significant jamming forces can only be developed as a result of the hoop reinforcement in the concrete.

3.1. Asphalt Corrosion Coating

Asphalt corrosion coating behaves as a visco-plastic material as illustrated by the test results from [2] reproduced in Figure 3.1. The comparison in [2] between the full scale bending tests to the analysis results lead to good agreement using a shear strength of 0.55MPa for rapid loading (Test 2), but for slower loading (over a period of 80min, Test 1) the good match was obtained for a shear strength of around 0.2 to 0.25MPa. This is consistent with the bond test results in Figure 3.1.

Thus for design based on the assumption of elastic perfectly-plastic behaviour, a bond strength around 0.5MPa seems appropriate for rapid loading (e.g. as the pipe enters the stinger), and 0.25MPa seems more appropriate for slower loading such as lateral buckling where it can be shown that dynamic[iii] buckling either does not occur, or occurs at a sufficiently low[iv] temperature.

The bond strength should be reduced for the effect of temperature. E.g. for slow loading at temperatures around 50°C a shear strength as low as 0.1MPa may be appropriate. Indeed as the temperature approaches the limit for application of asphalt of around 60°C to 70°C the bound strength can be expected to become very small, with the asphalt essentially providing lubrication rather than resistance.

3.2. Polyethene or Polypropylene Coating

The test results for polyethene (reproduced from [2] in Figure 3.2) show brittle rather than viscous behaviour. The maximum resistance is reached at small displacements (ranging from 0.1mm to 0.35mm), after which the shear stress drops rapidly to a residual value of 0.1MPa, "due mainly to friction" [2].

The questions that then arise are "At what point in the loading of the pipe will this brittle fracture of the bond occur?" and "Could significant strain concentrations in the steel develop before brittle fracture of the bond?" Fortunately in absence of slip the bond shear stress is fully concentrated at the point were the concrete stops, i.e. at the field joint. Therefore the bond is soon broken there and debonding can then propagate further into the concrete-coated portion of the pipe. Thus the shear stress for most of the interface will be at the residual value of 0.1MPa.

The above suggests that 0.1MPa is an appropriate value for use in design based on elastic-perfectly plastic behaviour. However it must be borne in mind that the frictional effects may not always be the same as for the tests performed in [2]. E.g. the normal force at the interface may increase due to pressurisation of the pipe, or jamming could occur as rings of concrete coating tend to twist with respect to the pipe, as illustrated in Figure 3.3. Indeed the full scale test in [2] showed a bit higher strain concentrations than those predicted with a shear strength of 0.1MPa (see Test 4, Figure 15 in [2]), but it is difficult to assess exactly to what extent this is due to the concrete cover, or due to local variations in steel strength.

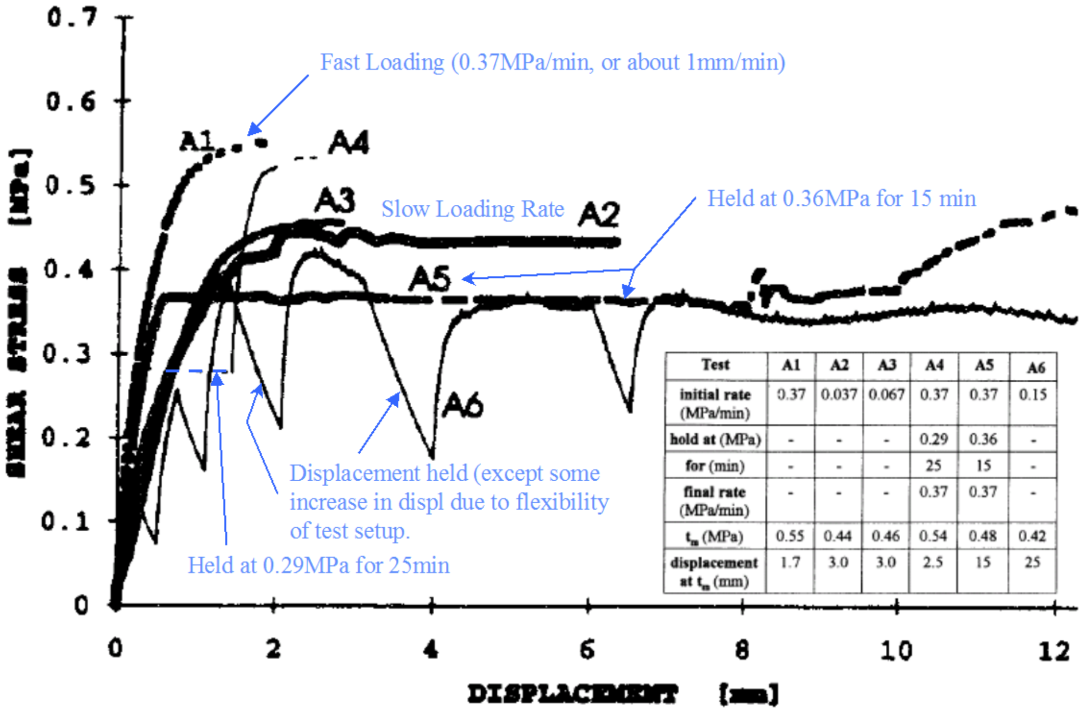
In view of the above 0.2MPa is considered to be a prudent design value for the bond strenght based on the assumption of elastic-perfectly plastic behaviour. This can reasonably be applied for polypropylene coating as well as polyethene. With further investigation involving an assessment of the mechanisms that lead to the residual frictional resistance a lower value may be justified.

In lieu of test data on the behaviour of the interface at higher temperatures, the behaviour of the interface between the polyethylene or polypropylene coating and the concrete may be assumed to be temperature independent. Indeed although the brittle resistance part of the bonding may be affected, this part of the resistance hardly affects the behaviour, and the later frictional resistance is less likely the be affected, unless the temperatures rise to the vicinity of the melting point for the polyethylene or polypropylene.

---

iii Dynamic buckling can occur if the initial imperfections to trigger buckling are small, so that the axial force rises considerably before buckling occurs.

iv There "low" means low enough so that the pipe deformations at this temperature will not be dangerous even for a high effective shear strength of the bond.



Test	A1	A2	A3	A4	A5	A6
initial rate (MPa/min)	0.37	0.037	0.067	0.37	0.37	0.15
hold at (MPa)	-	-	-	0.29	0.36	-
for (min)	-	-	-	25	15	-
final rate (MPa/min)	-	-	-	0.37	0.37	-
t_m (MPa)	0.55	0.44	0.46	0.54	0.48	0.42
displacement at t_m (mm)	1.7	3.0	3.0	2.5	15	25

Figure 3.1: Results of bond shear strength tests for asphalt corrosion coating at room temperature[v] from [2]. Shows shear stress on the interface (MPa) as a function of the relative displacement on the interface in mm. Different loading histories were applied, including monotonic tests at different rates (A1, A2, A3), creep tests in which the load was held constant (A4 and A5), and a relaxation test (A6) in which the displacement was held nominally constant (except for flexibility in the test setup) at various levels.

[v] The test temperature is not reported in [2], and is therefore presumed to be at room temperature.

Test	P1	P2	P3	P4
initial rate (MPa/min)	0.037	0.44	0.44	0.44
hold at (MPa)	-	-	0.18	0.18
for (min)	-	-	30	120
final rate (MPa/min)	-	-	0.44	0.44
t_m (MPa)	0.33	0.33	0.28	0.33
displacement at t_m (mm)	0.15	0.10	0.10	0.35
$t_{residual}$ (MPa)	0.10	0.10	0.10	0.10

Table 3: Results of tests on polyethene corrosion coating (t_m is the shear capacity and $t_{residual}$ the shear stress after fracture).

Figure 3.2: **Bond shear strength test results for polyethene corrosion coating from [2]. The bond stress drops rapidly from a maximum value t_m to a residual value of $t_{residual}$ = 0.1MPa, which is thought to be due to friction.**

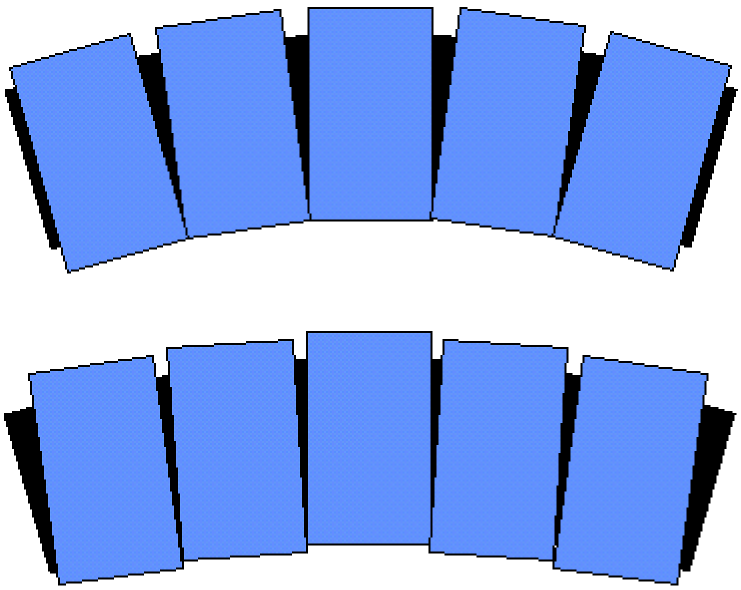


Figure 3.3: **Top: idealised deformation mechanism for concrete with plane perpendicular sections in the concrete remaining plane and perpendicular to the axis of the pipeline, but sliding axially along the pipeline because of the high stiffness of the concrete in compression relative to the bond strength. Bottom: actual behaviour – the rotation of the concrete sections does not quite match that of the pipe, so that axial sliding is more on the compression side that on the tension side. This type of behaviour is evident from Figure 12 of [2]. As a result of this, the concrete rings may have a tendency to jam, thereby increasing the effective axial resistance. This is because the frictional part of the resistance with rise with increasing normal force due to jamming**

REFERENCES

[1] Ness, O.B and Verley, R., (1995), "Strain Concentrations in Pipelines with Concrete Coating: An Analytical Model," Proc. OMAE'95, Copenhagen, Denmark.

[2] Verley, R., and Ness, O.B., (1995), “Strain Concentrations in Pipelines with Concrete Coating: Full Scale Bending Tests and Analytical Calculations,” Proc. OMAE 1995, Vol. 5, Pipeline Technology, ASME.

[3] Dafalias, Y.F. and Popov, E.P., (1976), "Plastic Internal Variable Formalism of Cyclic Plasticity," Journal of Applied Mechanics, Dec., pp. 645-651.

APPENDIX A. DESCRIPTION NPEX ELEMENT ROUTINES

The finite element calculations reported on here were performed by adding two new types of elements to a general finite element code. For each element type there is a set of two element routines: one by which material properties etc. are read and the storage requirements of the element are defined, and one to perform the actual calculations of the element contribution of the effective out-of-balance nodal forces and the element contribution to the tangent stiffness matrix. This Appendix describes the requirements for these two routines.

The routine to read material properties etc. is named "RELPROP_nn" where "nn" stands for a two digit number describing the element type, and the routine to perform the required calculations is named "GENELKP_nn". A description of the arguments for these routines and how they are to be used or defined is given in Tables A1 and A2.

Element groups are used, with all elements within a group being of the same type and having the same properties. These element properties are stored in an array AELPROP or IELPROP depending on whether they are real- or integer-valued. The RELPROP_nn routine is called once for every element group at the beginning of the analysis. Its function is the read the element properties. Typical element properties include material properties such at Young's modulus, the yield stress, and other material parameters as might be required by the material model used. They can be treated as element properties as long as the same values apply for all elements in the group. In contrast other quantities, such as the current stress in the element, or the current location and size of the yield surface vary from one element to the next within an element group. These will be referred to as "element parameters". They must be stored separately for each element and will typically also be updated at every loadstep once a converged solution has been obtained, before proceeding to the next loadstep.

The key output from the GENELKP routine are the out-of-balance forces ELP and the element tangent stiffness matrix ELK, both of which are assembled into corresponding global out-of-balance forces and the global tangent stiffness matrix used in the solution process by Newton iteration.

The program uses a simple user interface by which (unit 5) standard input is entered in response to prompts generated by the main program or the element routine RELPROP_nn. The input is then written to unit 9. One can then repeat the analysis without re-entering interactively all the input data, and make changes in some of the input parameters, if desired. Also if one has completed only part of an interactive input session, one can interrupt, correct values entered earlier as needed using the unit 9 file. Upon restarting the program will then first use that data available in unit 9, and then prompt the user interactively for the remainder of the required data. Details can be found by examining and of the ReaD RECord (RDREC) routines listed in Appendix C.

To describe the loading, load functions ALOADF may be used from within the element routine. Specifically ALOADF(TIME,i,n) provides the nth time derivative of the ith load factor as a function of a load parameter named TIME. In the analyses reported herein four such load functions were used: one for the initial curvature history to describe reeling, one for the submerged weight of the pipe, one for the internal pressure, and one for temperature. These load functions are defined outside the element routines by linear interpolation between user-specified values. The data describing the load functions are entered as part of the input data (outside the element routine), and stored in a common block. Therefore they need not pass through the element routines. Periodic load functions may also be used if one wants to examine ratcheting.

Table A.1: Arguments to subroutine RELPROP_nn.

Name	Description	Input/ Output
AELPROP	Array containing real-valued element properties to be read by the RELPROP routine either from unit 9 or from unit 5 depending on the value of IU9STAT.	O
IELPROP	Array containing integer valued element properties that are to be read by the RELPROP routine either from unit 9 or from unit 5 depending on the value of IU9STAT.	O
IEXFLAG	Contains information about type of analysis being performed, and may be needed only for special elements (e.g. to give an indication to read some additional element properties when needed)	I
IU9STAT	If IU9STAT=0 input data should be read from unit 9; otherwise they should be read from unit 5 and written to unit 9. If an attempt to read some input from unit 9 fails, then IU9STAT should be reset to a non-zero value and the required input should be read from unit 5 instead (preferably after prompting the user for the input), and written to unit 9. This should be done for every record that is read and has been achieved here by routines whose name starts with "RDREC", which are listed in Appendix C.	I & O
NAELDAT1	Maximum length of AELPROP (usually a very large number, but may be used to check that available storage for AELPROP is not exceeded).	I
NIELDAT1	Maximum length of IELPROP (usually a very large number, but may be used to check that available storage for IELPROP is not exceeded).	I
NEN	Number of nodes the element will be attached to.	O
NAELPAR	Number of real-valued *element parameters* used for each element. (Used to determine the storage requirement for real element parameters as NAELPAR times the number of elements in the element group.)	O
NAELPROP	Number of real element properties used (determines actual length of array AELPROP)	O
NDOF1	Number of degrees of freedom per node used by element. (E.g. 3 for 2D beam of pipe elements degrees of freedom consist of two components of dispacement plus a rotation.)	O
NIELPAR	Number of integer-values *element parameters* used for each element.	O
NIELPROP	Number of integer element properties used (determines actual length of array AELPROP)	O
NSD1	Number of coordinates per node used by element.	O

Table AA.2: Description and use of arguments to subroutine GENELKP_nn.

Variable or Array Name	Description	Input/ Output
AELPAR	Array of real-valued element parameters describing the effect of the prior deformation history of the element. Enters with the values of these parameters at the beginning of the loadstep. Depending upon the value of IELFLAG these may need to be updated.	I & O
AELPROP	Array of real-valued element properties.	I
ELK	Element stiffness matrix, dimensioned as ELK(NELDOF,NELDOF).	O
ELP	Array of element out-of-balance forces. When multiplied by corresponding virtual nodal displacements, one obtains EVW-IVW, where IVW and EVW are internal and external virtual work respectively.	O
ELP_L	Array containing the derivatives of ELP with respect to the load parameter TIME.	O
ID	Array for which ID(i,j) is the global degree of freedom number for nodal degree of freedom i at node j. Must be used to define the LM array, which describes the locations where the element out-of-balance forces ELP and element stiffness matrix ELK should be assembled into the corresponding global arrays for the entire system.	I
IELPAR	Similar to AELPAR, but for integer values element parameters.	I & O
IELPROP	Similar to AELPROP, but for interger-valued element properties.	I
IEN	Array for which IEN(i) is the global node number corresponding to element node number i.	I
LM	Integer array that describes the location where ELP and ELK is to be assembeled into the corresponding global arrays. Specifically LM(i) is the global degree of freedom number corresponding to element degree of freedom number I. This needs to be determined using the arrays ID and IEN.	O
U	Array containing the displacements at the end of the loadstep. Specifically U(i,j) is the displacement for nodal degree of freedom I at node j, where j is the global node number. [U(i,IEN(j)) is the displacement for nodal degree of freedom I at element node j.]	I
UB_L	Rate at which the specified portion of the displacements[vi] changes as a function of of the load parameter TIME. This is needed for	I

[vi] The total nodal displacements can be written as U=UB+Q, where UB represents the known portion of the nodal displacements which are specified as a function of time prior to beginning the analysis, and Q represents the unknown portion of the displacements. (As a result, Q is zero for those degrees of freedom where the displacements are specified, but on the other hand UB may be non-zero for degrees of freedom where the displacement is unknown as well as for those where the displacement is specified.) UB_L represents the derivative of UB with respect to the load parameter TIME. Furthermore ELP_L represents the derivative of ELP with respect to time for a fixed value of the unknown portion of the displacements Q. This means that for the purpose of calculating ELP_L, the derivatives of the nodal displacements with respect to TIME should be obtained from the array UB_L.

Variable or Array Name	Description	Input/ Output
	the calculation of ELP_L. Stored in the same format as the displacements U.	
U0	Array containing the displacements at the beginning of the loadstep in the same format as for the displacements U.	I
X	Array of nodal coordinates, can be used to calculate the geometric parameters of the element. Specifically X(i,j) corresponds to the ith nodal coordinate at node j, where j is the global node number. Note that the element may but need not make use of all NSD nodal coordinates that are present in the analysis.	I
IEL	Element identification number. Can be used when printing out stresses etc.	I
IELFLAG	Determines what tasks need to be performed by the GENELKP_nn routine. Specifically IELFLAG this the sum of the following quantities: 2 if ELK and ELP_L are to be calculated, 4 to update the element parameters AELPAR and IELPAR, 8 to write the stresses and other quantities of interest pertaining to the element to unit 6. Thus for instance if IELFLAG=0 only ELP, NELDOFI, and LM need to be calculated by the GENELKP_nn routine. If IELFLAG=14 then ELK and ELP_L need to be calculated as well, the element parameters need to be updated, and output for the element needs to be written to unit 6.	I
IGRP	Element group number. May be included in the output to unit 6 when appropriate.	I
ISTEP	Load step number.	I
NAELPAR	Length of array AELPAR.	I
NAELPRO P	Length of array AELPROP.	I
NDOF	Number of degrees of freedom per node present. (Not all these dof's need be used by the element.)	I
NDOF4BC	Used only by special elements, otherwise NDOF4BC=NDOF.	I
NELDOF	Maximum number of element degrees of freedom; determines declaration of arrays ELK, ELP, etc.	I
NELDOFI	Number of element degrees of freedom used by element; determines how many of the elements of ELK, ELP should be assembled into the global arrays.	O
NEN	Maximum number of nodes used by an element.	I
NIELPAR	Length of array IELPAR.	I
NIELPROP	Length of array IELPROP	I
NNODES	Total number of nodes present in the analysis.	I
NSD	Maximum number of nodal coordinates per node declared for the	I

Variable or Array Name	Description	Input/ Output
	analysis.	
TIME	Value of the load parameter at the end of the loadstep.	I
TIME0	Value of the load parameter at the beginning of the loadstep.	I

APPENDIX B. CONCRETE COATING ELEMENT IN FORTRAN

```
C 2-node, 2D Concrete Cover Element
C
C Deformation is in x-z plane.
C
C Nodal Coordinates are:
C 1) Axial x coordinate, and
C 2) Transverse z coordinate.
C
C Nodal Dof's are:
C 1) Axial u displacement,
C 2) Transverse w displacement (in z-direction), and
C 3) Rotation theta about y-axis (direction by RH rule).
C 4) Axial slip of concrete over pipe.
C
C Real Element Parameters (AELPAR) are:
C 1:NINTPT) ECMAX = max compressive strain experienced by the concrete
C NINTPT+1) VSPL(1) = axial plastic slip in tar interface at elmt node 1
C NINTPT+2) VSPL(2) = axial plastic slip in tar interface at elmt node 1
C
C Real Element Properties (AELPROP) are:
C 1) CDIA = inner diameter of concrete = OD of pipe
C 2) CTHK = concrete thickness
C 3) FPC = Concrete compressive strength
C 4) EPC = concrete strain at max compr stress
C 5) TAUY = tar interface shear strength
C 6) SLMOB = tar interface shear mobilisation slip
C 7) GGP = plastic hardening stiffness for shear-slip relation
C 8) EPST = thermal axial strain
C 9) SKAPPAT = thermal curvature
C 10) BFX = applied force per unit length in x direction
C 11) BFZ = applied force per unit length in z direction
C
C Integer Element Properties (IELPROP) are:
C 1) NINTPT = number of integration points around half circumference
C 2) ILFE = load function number for thermal axial strain
C 3) ILFK = load function number for thermal curvature
C 4) ILFBF = load function number for body forces

      SUBROUTINE RELPROP_22(AELPROP,IELPROP,     !arrays
C input/output -->           o      o
     &  IEXFLAG,IU9STAT,NAELDAT1,NIELDAT1,       !scalars
C        i      i&o     i       i
     &  NEN,NAELPAR,NAELPROP,NDOF1,NIELPAR,NIELPROP,NSD1) !scalars
C        o  o      o       o     o       o       o
C
      IMPLICIT REAL*8 (A-H,O-Z)
      DIMENSION IELPROP(NIELDAT1),AELPROP(NAELDAT1)
      WRITE(6,'(2X/
     &" == Concrete Cover Element (nelmt22) Chosen."/
     &"  NINTPT = # int pts around half cicumference")')
      CALL RDREC1I(NINTPT,'NINTPT',5,IU9STAT)
      IELPROP(1)=NINTPT
      NEN=2            !number of element nodes used
      NAELPAR=NINTPT+2  !number of real element parameters
      NAELPROP=11       !number of element properties needed
      NDOF1=4           !number of degrees of freedom per node needed
      NIELPAR=0         !number of integer element parameters
      NIELPROP=4        !number of integer element properties needed
      NSD1=2            !number of coordinates per node used by element
      CALL CHKINT(NAELPROP,NAELDAT1,'NAELPROP in nelmt22',19)
      CALL CHKINT(NIELPROP,NIELDAT1,'NIELPROP in nelmt22',19)
C
      WRITE(6,'(2X/
     &" CDIA = concrete inner diameter = OD of pipe"/
     &" CTHK = concrete thickness"/
     &" FPC = Concrete compressive strength"/
     &" EPC = concrete strain at max compr stress")')
      CALL RDRECNA(AELPROP(1),'CDIA,CTHK,FPC,EPC',17,IU9STAT,4)
      WRITE(6,'(2X/
     &" TAUY = tar interface shear strength"/
     &" SLMOB = tar interface shear mobilisation slip"/
     &" GGP = tangent stiffness for interface slip (kin. hard)")')
      CALL RDRECNA(AELPROP(5),'TAUY,SLMOB,GGP',14,IU9STAT,3)
```

```
C
      WRITE(6,'(2X/
     &" Initial axial strain and curvature are given by"/
     &"    EPST*ALOADF(TIME,ILFE,0), and"/
     &"    SKAPPAT*ALOADF(TIME,ILFK,0), respectively.")')
      CALL RDRECNA(AELPROP(8),'EPST,SKAPPAT',12,IU9STAT,2)
      CALL RDRECNI(IELPROP(2),'ILFE,ILFK',9,IU9STAT,2)
C
      WRITE(6,'(2X/
     &" Input body forces (per unit length) acting on element,"/
     &"   BFX = force per unit length in x-direction"/
     &"   BFZ = force per unit length in z-direction"/
     &"   ILFBF = time function number associated with body force, or"/
     &"          zero for constant body force acting at all times.")')
      CALL RDRECNA1I(AELPROP(10),IELPROP(4),'BFX,BFZ,ILFBF',
     & 13,IU9STAT,2)
      RETURN
      END
C End of subroutine RELPROP_22
C
C==============================================================================
====
C
      SUBROUTINE GENELKP_22(AELPAR,AELPROP,
     &  ELK,ELP,ELP_L,
     &  ID,IELPAR,IELPROP,
     &  IEN,LM,
     &  U,UB_L,U0,X,
     &  IEL,IELFLAG,IGRP,ISTEP,NAELPAR,NAELPROP,NDOF,NDOF4BC,
     &  NELDOF,NELDOFI,
     &  NEN,NIELPAR,NIELPROP,NNODES,NSD,TIME,TIME0)
C
      IMPLICIT REAL*8 (A-H,O-Z)
C
      DIMENSION AELPAR(NAELPAR),AELPROP(NAELPROP),
     & ELK(NELDOF,NELDOF),ELP(NELDOF),ELP_L(NELDOF),
     & ID(NDOF,NNODES),IELPAR(NIELPAR),IELPROP(NIELPROP),
     & IEN(NEN),LM(NELDOF),
     & U(NDOF,NNODES),UB_L(NDOF,NNODES),U0(NDOF,NNODES),X(NSD,NNODES)
C
C Local Declarations:
      LOGICAL LF_INDIC
      DIMENSION ELKB(6,6),IDB(8)
      DATA IDB/1,2,3,1,4,5,6,4/
      PARAMETER(PI=3.14159265359D0,DEG=0.01745329251994D0)
C
      DIMENSION UEL(8),UELB_L(8),D(3,3),DB(3,6),GGT(2),TAU(2)
C
C Initialise element parameters, if appropriate:
      IF(LF_INDIC(IELFLAG,6)) THEN
        IF(NAELPAR.GT.0) CALL CLEAR(AELPAR,NAELPAR)
        IF(NIELPAR.GT.0) CALL ICLEAR(IELPAR,NIELPAR)
        IF(IELFLAG.EQ.2**6) RETURN
      ENDIF
C
C Form LM,  UELB_L, and NELDOFI:
      J=0
      DO I=1,2
        INODE=IEN(I)
        DO II=1,4
          J=J+1
          LM(J)=ID(II,INODE)
          UEL(J)=U(II,INODE)
          UELB_L(J)=UB_L(II,INODE)
        ENDDO
      ENDDO
      NELDOFI=J
C
C Geometry & Displacement derivatives
      XX=0.5D0*(X(1,IEN(2))+X(1,IEN(1)))
      CL=X(1,IEN(2))-X(1,IEN(1))
      CZP=(  X(2,IEN(2))-X(2,IEN(1))  )/CL
      UP=(UEL(5)+UEL(8)-UEL(1)-UEL(4))/CL
      WP=(UEL(6)-UEL(2))/CL
      TP=(UEL(7)-UEL(3))/CL
      SZP=CZP+WP
```

```
C
C Recover Element Properties:
      CDIA=AELPROP(1)
      CTHK=AELPROP(2)
      FPC=AELPROP(3)
      EPC=AELPROP(4)
      TAUY=AELPROP(5)
      SLMOB=AELPROP(6)
      GGP=AELPROP(7)
      EPST=AELPROP(8)
      SKAPPAT=AELPROP(9)
      BFX=AELPROP(10)
      BFZ=AELPROP(11)
      NINTPT=IELPROP(1)
      ILFE=IELPROP(2)
      ILFK=IELPROP(3)
      ILFBF=IELPROP(4)
C
C Section Strains & time deriv for const nodal displ.
      EPS=UP+(CZP+0.5D0*WP)*WP
      SKAPPA=TP
      EPS=EPS-EPST*ALOADF(TIME,ILFE,0)
C     VS=0.5D0*(UEL(4)+UEL(8)) !relative axial slip
      EPS_L=-EPST*ALOADF(TIME,ILFE,1)
      SKAPPA=SKAPPA-SKAPPAT*ALOADF(TIME,ILFK,0)
      SKAPPA_L=-SKAPPAT*ALOADF(TIME,ILFK,1)
C
C Stress Resultants, CN, CM & derivatives D(i,j) wrt EPS & SKAPPA:
      CN=0.D0
      CM=0.D0
      CALL CLEAR(D,9)
      DTHETA=NINTPT-1
      DTHETA=PI/DTHETA
      RR=0.5D0*(CDIA+CTHK)
      TRXSA1=RR*DTHETA*CTHK
      TRXSA2=TRXSA1+TRXSA1
      EEC=2.D0*FPC/EPC !elastic modulus of concrete
c     EPC2=EPC+EPC
      DO I=1,NINTPT
        THETA=I-1
        THETA=THETA*DTHETA  !theta = 0 at 12 o'clock
        ZZ=RR*COS(THETA)
        EPSX=EPS+ZZ*SKAPPA
        ECMAX0=AELPAR(I)
        ECMAX1=MAX(ECMAX0,-EPSX)
C   Calc conc stress (SC), tangent modulus (ETC):
        IF(EPSX.GT.0.D0) THEN !cracks open
          SC=0.D0
          ETC=0.D0
        ELSE IF(ECMAX1.GT.ECMAX0) THEN
          CALL SECONC1(FCMAX1,ETC,ECMAX1,FPC,EPC,EEC)
          SC=-FCMAX1
          IF(LF_INDIC(IELFLAG,2)) AELPAR(I)=ECMAX1
        ELSE
          CALL SECONC1(FCMAX0,ETC,ECMAX0,FPC,EPC,EEC)
          SCEL=-FCMAX0+EEC*(EPSX+ECMAX0)
          IF(SCEL.GT.0.D0) THEN
            SC=0.D0
            ETC=0.D0
          ELSE
            SC=SCEL
            ETC=EEC
          ENDIF
        ENDIF
C
        IF(LF_INDIC(IELFLAG,3)) THEN
          WRITE(6,'(" X,THETA,EPSX,SC,ECMAX",5G13.5)')
     &          XX,THETA/DEG,EPSX,SC,ECMAX1 !theta in deg
        ENDIF
        IF(I.EQ.1.OR.I.EQ.NINTPT) THEN
          SC=SC*TRXSA1
          ETC=ETC*TRXSA1
        ELSE
          SC=SC*TRXSA2
          ETC=ETC*TRXSA2
        ENDIF
```

```
         CN=CN+SC
         CM=CM+SC*ZZ
         D(1,1)=D(1,1)+ETC
         TST1=ETC*ZZ
         D(1,2)=D(1,2)+TST1
         D(2,2)=D(2,2)+TST1*ZZ
        ENDDO
        D(2,1)=D(1,2)
        CN_L=D(1,1)*EPS_L+D(1,2)*SKAPPA_L
        CM_L=D(2,1)*EPS_L+D(2,2)*SKAPPA_L
C
C Calc shear stress in tar (TAU), tangent shear stiffness (GGT); update VSPL:
C   The basis for this is a kinematic hardening model in which
C   TAU=GGE (VS-VSPL)
C   TAUyield=GGP*VSPL+/-TAUY
C
        GGE=TAUY/SLMOB            !elastic stiffness
        DO I=1,2
         VSPL=AELPAR(NINTPT+I)     !plastic slip
         VS=UEL(4*I)
         TAUE=GGE*(VS-VSPL)        !elastic predictor stress
         TAUSH=GGP*VSPL
         TAUYP=TAUSH+TAUY          !yield stress for the positive direction
         TAUYN=TAUSH-TAUY          !yield stress for the negative direction
         IF(TAUE.GT.TAUYP) THEN    !yielding in the positive direction
          GGT(I)=GGP*GGE/(GGE+GGP)     !tangent stiffness
          TAU(I)=TAUYP+GGT(I)*(TAUE-TAUYP)/GGE    !current stress
          VSPL=VS-TAU(I)/GGE                  !current plastic slip
         ELSE IF(TAUE.LT.TAUYN) THEN  !yielding in the negative direction
          GGT(I)=GGP*GGE/(GGE+GGP)     !tangent stiffness
          TAU(I)=TAUYN+GGT(I)*(TAUE-TAUYN)/GGE    !current stress
          VSPL=VS-TAU(I)/GGE                  !current plastic slip
         ELSE                      !elastic behaviour
          TAU(I)=TAUE
          GGT(I)=GGE
         ENDIF
         IF(LF_INDIC(IELFLAG,2)) AELPAR(NINTPT+I)=VSPL
        ENDDO
C
        TRAREA=0.5D0*PI*CDIA*CL
C
C Compute Element out-of-balance load vector:
        CQ1=SZP*CN
        TST=0.5D0*CL*ALOADF(TIME,ILFBF,0)
        BFX1=TST*BFX
        BFZ1=TST*BFZ
        ELP(1)=CN+BFX1
        ELP(2)=CQ1+BFZ1
        ELP(3)=CM
        ELP(4)=ELP(1)-TAU(1)*TRAREA
        ELP(5)=-CN+BFX1
        ELP(6)=-CQ1+BFZ1
        ELP(7)=-CM
        ELP(8)=ELP(5)-TAU(2)*TRAREA
C
        IF(LF_INDIC(IELFLAG,1)) THEN  !compute ELK and ELP_L
C
C   Form DB(I,J) =  SUM(  D(I,K) B(K,J)  ,K=1,3)
         DO I=1,3
          TST1=D(I,1)
          TST2=SZP*D(I,1)+D(I,3)
          TST3=D(I,2)
          TST1=TST1/CL
          TST2=TST2/CL
          TST3=TST3/CL
          TST4=D(I,3)/2.D0
          DB(I,1)=-TST1
          DB(I,2)=-TST2
          DB(I,3)=-TST3+TST4
          DB(I,4)=+TST1
          DB(I,5)=+TST2
          DB(I,6)=+TST3+TST4
         ENDDO
C
C Form ELKB(I,J) = SUM(  B(K,I)*DB(K,J)   ,K=1,3)
         DO J=1,6
```

```
          TST1=DB(1,J)
          TST2=SZP*DB(1,J)+DB(3,J)
          TST3=DB(2,J)
          TST4=DB(3,J)*CL/2.D0
          ELKB(1,J)=-TST1
          ELKB(2,J)=-TST2
          ELKB(3,J)=-TST3+TST4
          ELKB(4,J)=+TST1
          ELKB(5,J)=+TST2
          ELKB(6,J)=+TST3+TST4
         ENDDO
C Add geometric stiffness matrix
         TST1=CN/CL
         ELKB(2,2)=ELKB(2,2)+TST1
         ELKB(2,5)=ELKB(2,5)-TST1
         ELKB(5,2)=ELKB(5,2)-TST1
         ELKB(5,5)=ELKB(5,5)+TST1
C Transform to new dof's & add contribution from slip stiffness:
         DO I=1,8
          IB=IDB(I)
          DO J=1,8
           JB=IDB(J)
           ELK(I,J)=ELKB(IB,JB)
          ENDDO
         ENDDO
         ELK(4,4)=ELK(4,4)+GGT(1)*TRAREA
         ELK(8,8)=ELK(8,8)+GGT(2)*TRAREA
C Compute ELP_L:
         CQ1=SZP*CN_L
         TST=0.5D0*CL*ALOADF(TIME,ILFBF,1)
         BFX1=BFX*TST
         BFZ1=BFZ*TST
         ELP_L(1)=CN_L+BFX1
         ELP_L(2)=CQ1+BFZ1
         ELP_L(3)=CM_L
         ELP_L(4)=ELP_L(1)
         ELP_L(5)=-CN_L+BFX1
         ELP_L(6)=-CQ1+BFZ1
         ELP_L(7)=-CM_L
         ELP_L(8)=ELP_L(5)
         DO I=1,NELDOFI
          TST1=ELP_L(I)
          DO J=1,NELDOFI
           TST1=TST1-ELK(I,J)*UELB_L(J)
          ENDDO
          ELP_L(I)=TST1
         ENDDO
        ENDIF
        IF(LF_INDIC(IELFLAG,3)) THEN !print stresses in element
         WRITE(6,1100) IEL,XX,CN,CM
         WRITE(6,1200) X(1,IEN(1)),(-ELP(I),I=1,4)
         WRITE(6,1300) X(1,IEN(2)),(-ELP(I),I=5,8)
        ENDIF
        RETURN
1100  FORMAT(' IEL=',I7,'  X=',G12.4,'  N=',G12.4,'  M=',G12.4)
1200  FORMAT(' X=',G12.4,'  S1:S4=',4G12.4)
1300  FORMAT(' X=',G12.4,'  S5:S8=',4G12.4)
        END
C
        SUBROUTINE SECONC1(FC,ETC,EC,FPC,EPC,EEC)
C   input/output         o  o  i  i   i   i
C Defines backbone stress-strain relation for concrete in compression.
C Parabolic stress-strain relation up to ultimate; const stress thereafter.
C FC = concrete stress (positive for compression)
C ETC = tangent modulus
C EC = strain in concrete (positive for compression)
C FPC = concrete compressive strength
C EPC = concrete strain at ultimate compression (positive)
C EEC = 2 FPC / EPC = concrete elastic modulus
        IMPLICIT REAL*8 (A-H,O-Z)
        XI=(EPC-EC)/EPC
        IF(XI.GT.0.D0) THEN
         FC=FPC*(1.D0-XI*XI)
         ETC=EEC*XI
        ELSE
         FC=FPC
```

```
         ETC=0.D0
       ENDIF
       RETURN
       END
C
C Log of Developments
C Derived from nelmt18 (mod defl elastic beam element) on 2 Oct 2001.
C nelmt18 appears to have been derived from nelmt04 (mod defl pipe element)
C 12 Feb 2006: adding a small stiffness against slip, because of convergence
C   difficulties encountered without this when none of the slip dof's within
C   a pipe joint are restrained. Input files require adding GGT after SLMOB
C   (on same line) as part of the element property input.
C 12 Feb 2006: changing number of integration points for slip work from
C   1 in the middle of the element to 2, with one at each node.
C   Otherwise zero energy deformation modes form when all the concrete
C   is cracked in tension.  This rather than the lack of stiffness
C   against slip seems to have been the reason for convergence difficulties.
```