FS2000

Pipeline Analysis
Properties
Exanpansion
UHB
Pipe Walking

Advanced Structural Analysis
for Windows
(c) A.E.S. Ltd 1988,2015

Email:support@aes-uk.com

Table of Contents

Contents **3**

1.0 Introduction 3
2.0 Operation 4
3.0 Surface Pipeline 6
4.0 Partially Buried Pipeline 8
5.0 Buried Pipeline 10
5.1 Axial Stiffness 12
5.2 Lateral Stiffness 13
5.3 Vertical Downward Stiffness 15
5.4 Vertical Upward Stiffness 16
5.5 Buried Pipeline with a vertical breakout model (Upheaval buckling) 17
6.0 UHB Model Generator 18
6.1 UHB Definition Data 20
6.1 Model Description 23
6.2 Load Cases & Combination 25
6.3 Time Curves & DyNoFlex Options 27
6.4 Extracting Results from a UHB Model 29
7.0 Expansion Analysis 32
7.1 Expansion Defintion Data 33
8.0 Reeling Model 35
8.1 Reeling Model Definition Data 36

Contents

1.0 Introduction

This utility program/wizard will generate the basic analysis properties, both stiffness and foundation properties, for a variety of pipeline design configurations. The generated data includes general pipeline properties and specific command line instructions which define element properties in FS2000.

The purpose of the utility is to:

- Create suitable geometric pipe and soil support analytical properties for FS2000 based on specific pipeline engineering configuration data (Section 2)
- Create complete FS2000 models for pipeline expansion analysis (Section 7)
- Create complete FS2000 models for pipeline sequential expansion analysis - pipe walking (Section 7)
- Create complete FS2000 models for pipeline upheaval buckiling analysis (Section 6)

The evaluation of properties are based on the ASCE's "Guideline for the Design of Buried Steel Pipe", 2001 or common industry practice.

The entered data can be saved in property tables in the same manner as properties in FS2000. When the data is saved it is saved with the current default model.

The output from this utility program is saved as a 'UM*' model file. This file is a formatted data file showing all input data and generated properties and will be included in the FS2000 model formatted data output. The file also includes the property data command lines. These commands enable the generated command line data to be interpreted within the Model Definition TASK of FS2000.

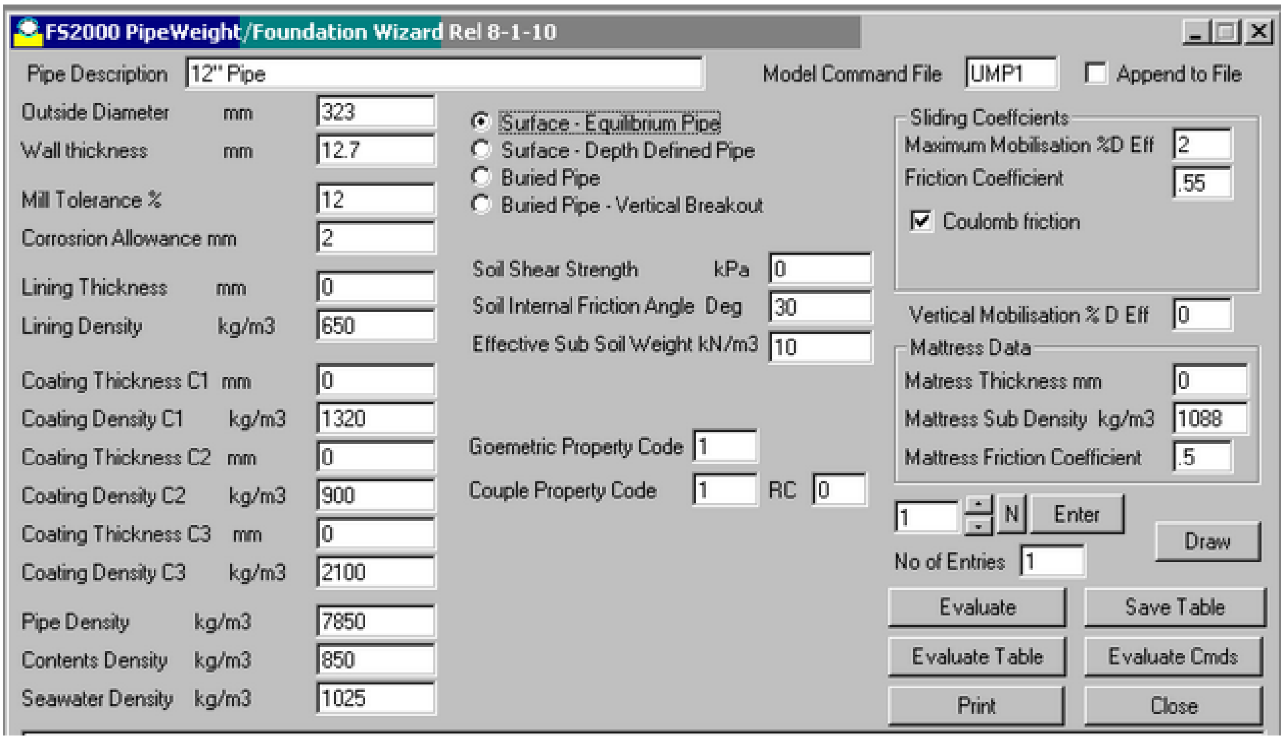
.

.

-o-

2.0 Operation

The program operates from one main window. When the program is started the current model pipe property table is loaded (if one exists).



Pipe data is entered in the main form shown below. Most of the pipe data is self descriptive, the use of less obvious data will be described in the later sections.

The basic operation is to define the data and then use the **Evaluate** button to generate the derived data. The derived data will appear in the panel and the data will be saved as a 'UM*' model file. The actual ID for the UM file can be changed using the **Model Command File** input box. The **Append to File** option can be used append the data to an existing data file.

The **Draw** button is used to show the pipe relative the the foundation surface.

The program generates properties for the following basic pipeline configurations:

- Surface Pipeline
- Partially Buried Pipeline
- Buried Pipeline
- Buried Pipeline with a vertical breakout model (Upheaval buckling)

Creating a Pipe Table

Pipe property data can be saved in a model dependent table in similar manner to property code data in FS2000.

On completion of data entry the **Enter** button is used to enter the data into the flange table. This is repeated for as many pipes that are required to be processed. Each time the Enter button is pressed the current data will be assigned to the current Pipe Table ID Number. Up to 20 pipe entries can exist in a pipe table.

The **Save Table** button is used to save the table entries to the current FS2000 model (text file called **<modelname>.UMPipeProp**).

The **Evalaute Table** button can be used to evalaute all the pipes in the pipe table.

The **Evalute Cmds** button an be used to evalaute all the pipes in the pipe table but will restrict the output to only the FS2000 command data.

-o-

3 Surface Pipeline

3.1 Soil Bearing Model

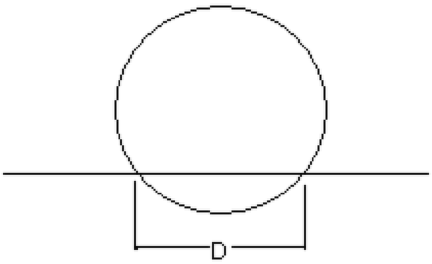
This foundation model represents a pipeline at its equilibrium position when resting on a cohesive or non-cohesive soil.

The above is used to define the soil properties.

The bearing capacity is based on the following well known foundation model:

$$Q_d = N_c cD + N_q \gamma HD + N_\gamma \gamma \frac{D^2}{2}$$

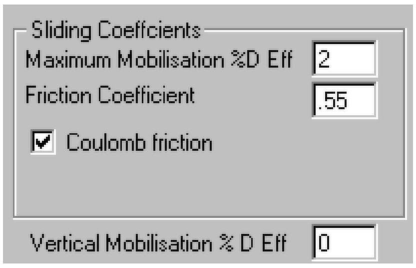
D is the effective bearing width based on the equilibrium depth of penetration of pipe into the soil.
H is the depth of penetration.
The bearing factors are given in Section 5.5



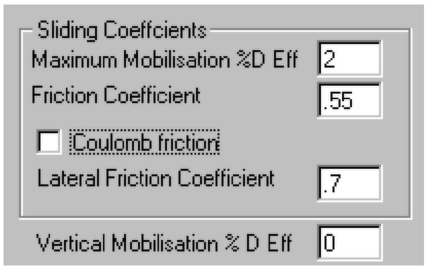
3.2 Friction

Sliding Resistance is given by Rs = Qd / μs

Coulomb Friction



Different Axial and Lateral Coefficients



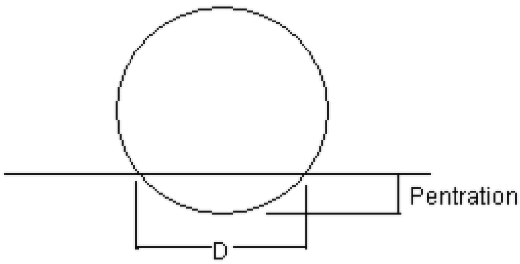
3.2 Stiffness

The horizontal sliding stiffness is defined by the **Maximum Mobilisation % D Eff** and Rs the sliding resistance due to friction. For surface pipes **D Eff** is taken to be the effective bearing width and not the outside diameter. When the **Coulomb friction** option is not active the and two friction coefficients are defined the sliding stiffness will be based on the direction with the highest resistance.

The vertical stiffness is defined by Kv = Qd / (Settlement Adjustment + Penetration) where,

Settlement Adjustment is based on the **Vertical Mobilisation %D** (using the effective bearing width D)
Penetration is the evaluated equilibrium penetration based on the the effective on bottom pipe weight and bearing width D to support it.
If **Vertical Mobilisation %D** is set to zero the vertical stiffness will be set by the equilibrium penetration depth of penetration only.

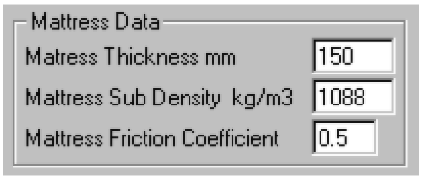
The **Vertical Mobilisation** would normally be set to zero but can be adjusted to give a desired mobilsation accounting for calculated penetration.



Non-linear vertical stiffness

The ultimate bearing resistance as a function of penetration depth is evaluated at penetration depths of (% of Eff OD) 0.5, 1.5, 5, 25, 10, 100 & 10000. If the **RC Property** Code box is entered as a non-zero value then the penetration stiffness will be defined in terms of the RC constantsin the table and can be used to represent a non-linear vertical stiffness in the contact elements. This can useful in cases where the pipe is pushed into the soil i.e. when large vertical displacement are expected.

3.4 Matress Effects



The matress is assumed to have an active bearing weight (Wm) on the top of the pipe equivalent to an effective width of 2.5D, where D is the effective outside diameter of the pipe. This load which should be applied to the model is listed in the output.

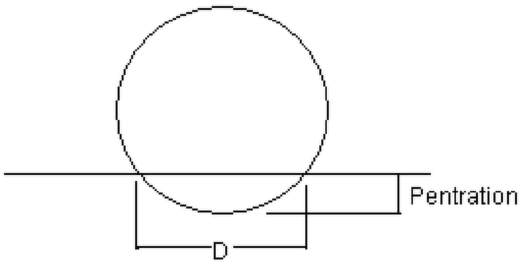
The Mattress Friction Coefficient μm is assumed to act between the top of the pipe and the mattress

The effective friction coefficient is evaluated from $\mu = Wm.\mu m + (Wm + Wp).\mu s / (Wm + Wp)$

-o-

4.0 Partially Buried Pipeline

This option is use is used to represent a pipe which is partially buried to a known depth. The effective bearing width is based on the depth of penetration. The maximum bearing width is the effective outside diameter of the pipe.



4.1 Soil Bearing Model

This foundation model represents a pipeline at its equilibrium position when resting on a cohesive or non-cohesive soil.

The above is used to define the soil properties.

The bearing capacity is based on the following well known foundation model:

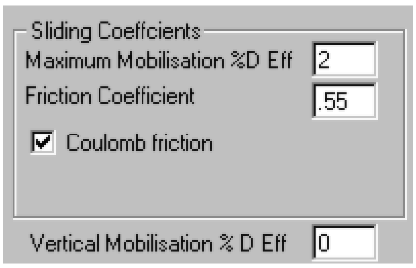
$$Q_d = N_c c D + N_q \gamma H D + N_\gamma \gamma \frac{D^2}{2}$$

H is the depth of penetration.

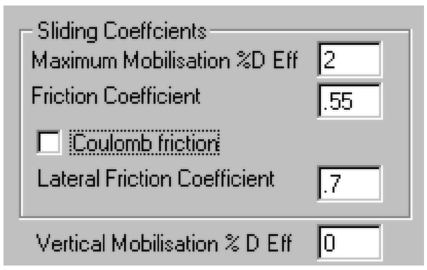
4.2 Friction

Sliding Resistance is given by Rs = Qd / μs

Coulumb Friction



Different Axial and Lateral Coefficients

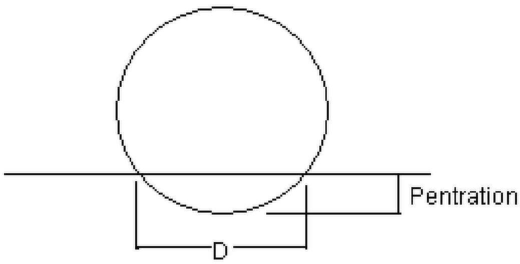


4.2 Stiffness

The horizontal sliding stiffness is defined by the **Maximum Mobilisation % D Eff** and Rs the sliding resistance due to friction. When the **Coulomb friction** option is not active the and two friction coefficients are defined the sliding stiffness will be based on the direction with the highest resistance.

The vertical stiffness is defined by Kv = Q_d / (**Vertical Mobilisation %D Eff)** (Typically 5-10%).

For surface penetrating pipes **D Eff** is taken to be the effective bearing width and not the outside diameter.



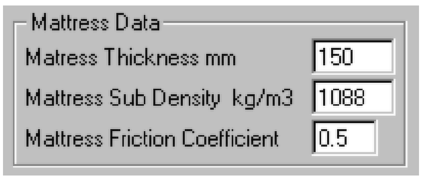
Non-linear vertical stiffness

This can be used in cases where the pipe is pushed into the soil i.e. when large vertical displacement are expected.

The ultimate bearing resistance is a function of penetration depth is evaluated at penetration depths of (% of Eff OD) 0.5, 1.5, 5, 25, 10, 100 & 10000.

If the **RC Property** Code box is entered as a non-zero value then the penetration stiffness will be defined in terms of the RC constants in the table and can be used to represent a non-linear vertical stiffness in the contact elements.

3.4 Mattress Effects



The mattress is assumed to have an active bearing weight (Wm) on the top of the pipe equivalent to an effective width of 2.5D, where D is the effective outside diameter of the pipe. This load which should be applied to the model is listed in the output.

The Mattress Friction Coefficient μm is assumed to act between the top of the pipe and the mattress

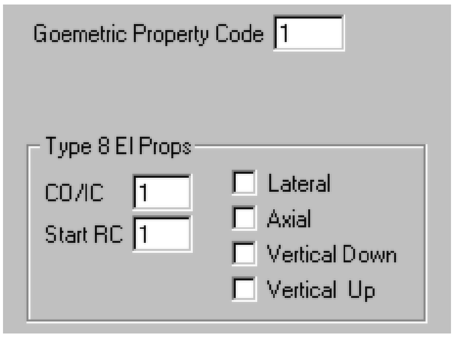
The effective friction coefficient is evaluated from $\mu = Wm.\mu m + (Wm + Wp).\mu s / (Wm + Wp)$

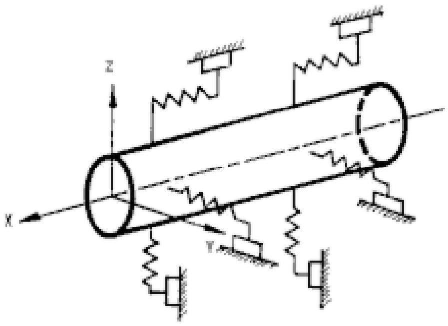
-o-

5.0 Buried Pipeline

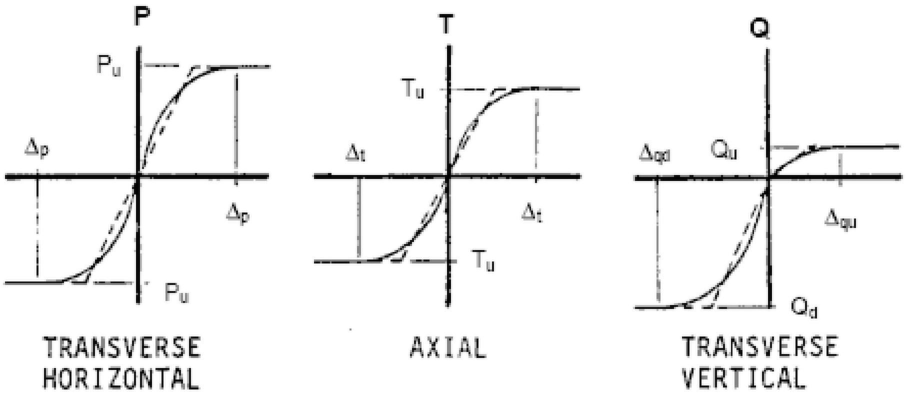
The evaluation of properties are based on the ASCE's "Guidleline for the Design of Buried Steel Pipe", 2001.

The stiffness properties generated are for use with Type 8 Elements. The bi-linear spring data is stored in the RC constant table.





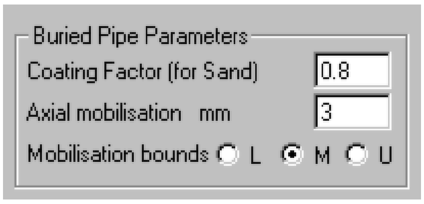
b) Idealized Representation of Soil with Discrete Springs



-o-

5.1 Axial Stiffness

The Coating Factor and the Axial mobilisation distances are defined below.



The maximum axial soil force per unit length of pipe that can be transmitted to the pipe is:

$$T_U = \pi D \alpha c + \pi D H \gamma \frac{1+K_0}{2} \tan\delta$$

where:

- D = pipe outside diameter
- c = soil cohesion representative of the soil backfill
- H = depth to pipe centerline
- γ = effective unit weight of soil
- K_o = coefficient of pressure at rest
- α = adhesion factor
- $\alpha = 0.608 - 0.123c - \dfrac{0.274}{c^2+1} + \dfrac{0.695}{c^3+1}$ where c is in ksf or kPa/100
- δ = interface angle of friction for pipe and soil = $f\phi$
- ϕ = internal friction angle of the soil
- f = coating dependent factor relating the internal friction angle of the soil to the friction angle at the soil-pipe interface

Representative values of *f* for various types of external pipe coatings are provided in the following table:

Pipe Coating	f
Concrete	1.0
Coal Tar	0.9
Rough Steel	0.8
Smooth Steel	0.7
Fusion Bonded Epoxy	0.6
Polyethylene	0.6

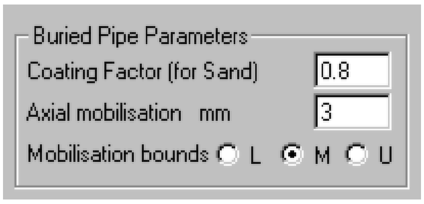
Friction factor f for Various External Coatings

Δ_t = displacement at T_u
= 0.1 inches (3 mm) for dense sand
= 0.2 inches (5 mm) for loose sand
= 0.3 inches (8 mm) for stiff clay
= 0.4 inches (10 mm) for soft clay

-o-

5.2 Lateral Stiffness

The **Mobilisation bounds** sets the displacement value between upper, lower and mid.



The maximum lateral soil force per unit length of pipe that can be transmitted to the pipe is:

$$P_u = N_{ch} cD + N_{qh} \gamma HD$$

where:

N_{ch} = horizontal bearing capacity factor for clay (0 for $c = 0$)

N_{qh} = horizontal bearing capacity factor (0 for $\phi = 0°$)

The expressions below for N_{ch} and N_{qh} are closed form fits to published empirical (plotted) results (see Figure B.3).

N_{ch} = horizontal bearing capacity factor for clay (0 for $c = 0$)

$$= a + bx + \frac{c}{(x+1)^2} + \frac{d}{(x+1)^3} \leq 9$$

N_{qh} = horizontal bearing capacity factors for sand (0 for $\phi = 0°$)

$$= a + b(x) + c(x^2) + d(x^3) + e(x^4)$$

Factor	ϕ	x	a	b	c	d	e
N_{ch}	0°	H/D	6.752	0.065	-11.063	7.119	--
N_{qh}	20°	H/D	2.399	0.439	-0.03	$1.059(10)^{-3}$	$-1.754(10)^{-5}$
N_{qh}	25°	H/D	3.332	0.839	-0.090	$5.606(10)^{-3}$	$-1.319(10)^{-4}$
N_{qh}	30°	H/D	4.565	1.234	-0.089	$4.275(10)^{-3}$	$-9.159(10)^{-5}$
N_{qh}	35°	H/D	6.816	2.019	-0.146	$7.651(10)^{-3}$	$-1.683(10)^{-4}$
N_{qh}	40°	H/D	10.959	1.783	0.045	$-5.425(10)^{-3}$	$-1.153(10)^{-4}$
N_{qh}	45°	H/D	17.658	3.309	0.048	$-6.443(10)^{-3}$	$-1.299(10)^{-4}$

N_{qh} can be interpolated for intermediate values of ϕ between 20° and 45°.

Δ_p = displacement at P_u

$$= 0.04\left(H + \frac{D}{2}\right) \leq 0.10D \text{ to } 0.15D$$

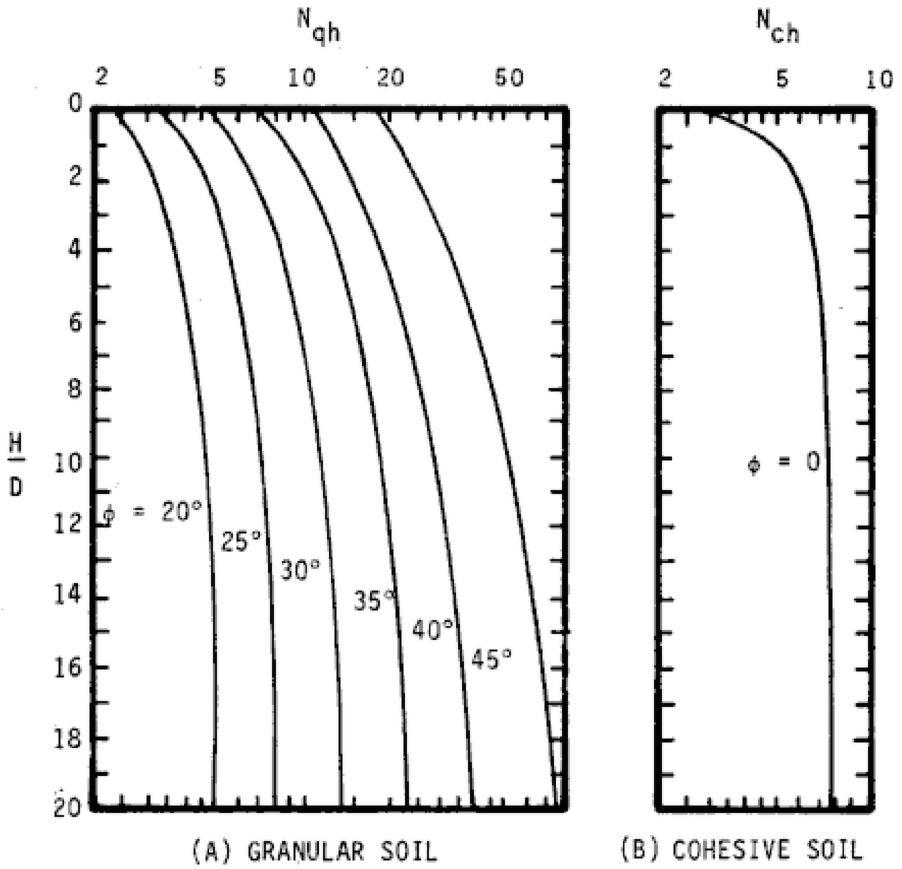


Figure B.3 Values of Nqh *and* Nch *of Hansen 1961*

-o-

5.3 Vertical Downward Stiffness Stiffness

$$Q_d = N_c cD + N_q \gamma HD + N_\gamma \gamma \frac{D^2}{2}$$

where:

N_c, N_q, N_γ	=	bearing capacity factors
N_c	=	$[\cot(\phi + 0.001)]\{\exp[\pi \tan(\phi + 0.001)] \tan^2\left(45 + \frac{\phi + 0.001}{2}\right) - 1\}$
N_q	=	$\exp(\pi \tan \Phi) \tan^2\left(45 + \frac{\phi}{2}\right)$
N_γ	=	$e^{(0.18\phi - 2.5)}$
γ	=	total unit weight of soil
Δ_{qd}	=	displacement at Q_d
	=	$0.1D$ for granular soils
	=	$0.2D$ for cohesive soils

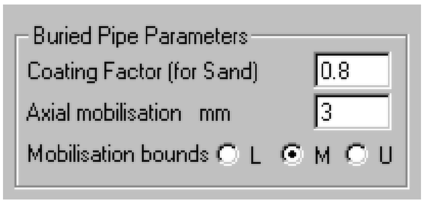
In the program the following expressions are used to evaluate the bearing factors (ref. API RP2A 21Ed)

N_q =	(Exp [π tanϕ]) ($\tan^2(45° + \phi'/2)$), a dimensionless function of ϕ',
N_c =	$(N_q - 1)\cot\phi'$, a dimensionless function of ϕ',
N_γ =	an empirical dimensionless function of ϕ' that can be approximated by $2(N_q + 1)\tan\phi$,

-o-

5.4 Vertical Upward Stiffness Stiffness

The **Mobilisation bounds** set the displacement value between upper, lower and mid.



The equations for determining upward vertical soil spring forces are based on small-scale laboratory tests and theoretical models. For this reason, the applicability of the equations is limited to relatively shallow burial depths, as expressed as the ratio of the depth to pipe centerline to the pipe diameter (*H/D*). Conditions in which the *H/D* ratio is greater than the limit provided below require case-specific geotechnical guidance on the magnitude of soil spring force and the relative displacement necessary to develop this force.

$$Q_u = N_{cv} cD + N_{qv} \gamma HD$$

where:

N_{cv} = vertical uplift factor for clay (0 for $c = 0$)

N_{qv} = vertical uplift factor for sand (0 for $\phi = 0°$)

$N_{cv} = 2\left(\frac{H}{D}\right) \le 10$ applicable for $\left(\frac{H}{D}\right) \le 10$

$N_{qv} = \left(\frac{\phi H}{44D}\right) \le N_q$ (See Section B.4 for definition of N_q)

The above equations represent an approximation to published results

Δ_{qu} = displacement at Q_u

= $0.01H$ to $0.02H$ for dense to loose sands < 0.1D

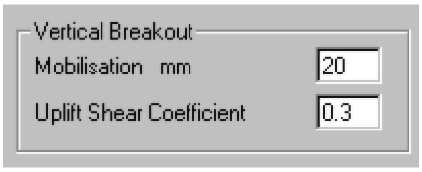
= $0.1H$ to $0.2H$ for stiff to soft clays < 0.2D

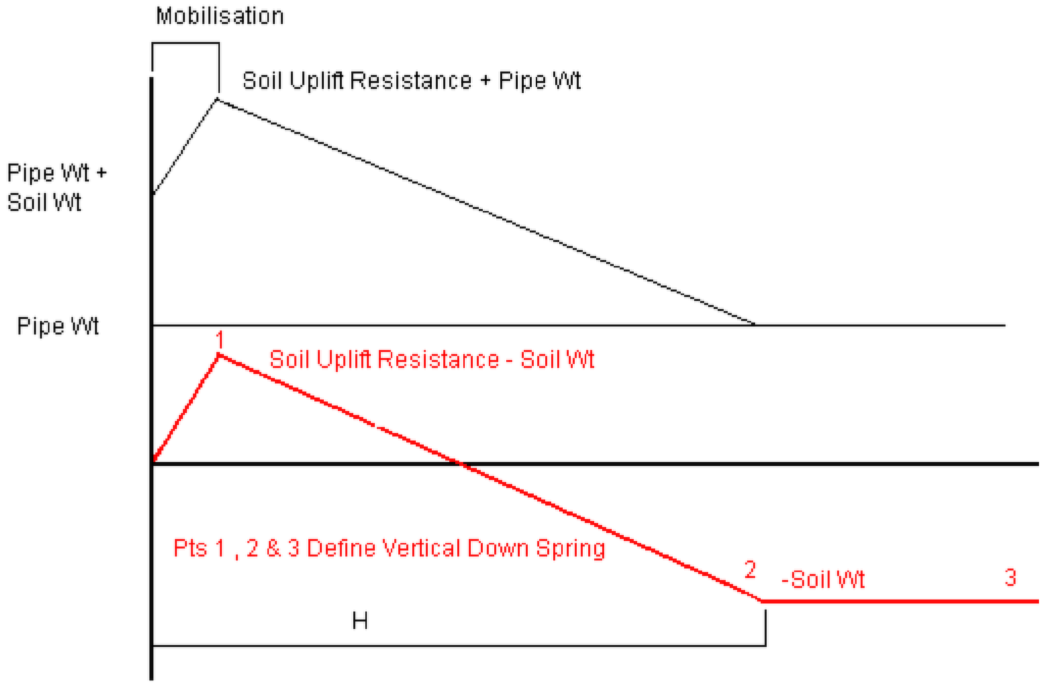
-o-

5.5 Buried Pipeline (Upheaval buckling)

When this option is selected the stiffness of the soils are as those for the buried pipeline apart from the vertical upward direction. In this direction a breakout resistance model is used. The only additional input required are **Mobilisation** and **Uplift Shear Coefficient.**

The output will define the magnutude of the UDL to represent the soil submerged weight resistance.





For Sand

Soil Uplift Resistance = γ.D.H.(1 + f.H/D)

For Clay

Soil Uplift Resistance = γ.D.H + 2.H.c

The Soil Wt = γ.D.H

The output will define the magnutude of the UDL to represent the submerged soil weight resistance (γ.D.H).

-o-

6.0 UHB Model Generator

The UHB Model Generator is a utility that will generate a complete UHB model including all load cases, combinations, batch files and solution option data. The results obtained from this model are in the form of design curves showing the effects of cover heights and imperfections on axial buckling force.

The program also generates a batch file that can be used to evaluate the weight requirements ie the downward force to prevent UHB for given imperfection profiles. It does this by using monitoring springs to measure the uplift loads generated for a given temperature case. This can be useful for investigating the extent of cover required for crossing or specific seabed features.

The main features of the utility are:

- All pipe properties and seabed stiffness are generated from pipe data input form
- Single pipe and Pipe-in-Pipe models can be generated
- Multiple 'Prop' type imperfection profiles with varying wavelength factors
- Defined Crossing profiles
- Full model or symmetrical model

The generated model consists of 3 basic zones:

- Feed in Zone - Region of coarse Type 8 elements
- Intermediate Zone - Region of intermediate Type 8 elements
- Imperfection Zone - Fine mesh of Type 8 elements

A more detailed description of the generated model is given in Section 6.1.

The following basic procedure should be followed to create the model and complete the analysis

1. Open a new model. Do not generate any data.
2. Start the Pipe Properties Utility
3. Use the Pipe Properties to define the pipe and foundation properties and save the table. The **Buried Pipe Vertical Breakout** option must be active.
4. Click the **UHB Model** button to make the UHB data form visible
5. Enter the UHB design parameters appropriate to the pipeline configuration.
6. Click the **Generate Model Data (UMUHB File)** button.
7. Return to the FS2000 GUI
8. In the Model Definition TASK use the Interpret File command to interpret the <Model>.UMUHB file.
9. Save the model
10. Go to the Primary TASK and Archive the model.
11. Open the model from its Archive format.
12. Open the batch file(**<model>.BRM)** and run it.
13. On completion of the batch run all results should exist.

To run the downward form monitoring analysis open the **<model>.BRM_Mon** batch file and execute the batch.

Generated Pipe Properties

- ***Geometric Properties***

For rigid pipe there should be no reason to change the properties of the generated model. If however non standard pipe e.g. flexible pipe is being assessed then the geometric properties will require to modified in the FS2000 GUI to give the required EA and EI properties.

For flexible pipe the following is required to be done:

  - Input the OD and overall wall thickness of the pipe and its effective density in the pipe properties form
  - Enter the E value and I value inthe UHB form to give the correct EA and EI i.e E = EA / Actual Pipe Area and I = EI / E
  - After interpretation, redefine I in the FS2000 property input box. To do this enter the OD and t, then change the I value to make ii a USD section. Note the OD and t have to be those of the

pipe so that the end cap force is correctly evaluated.
- Enter the value of E in the material property box **-** The value of Poisson's ratio should also be set to suit the type of pipe

- Material Properties

The generated material properties are for steel X65 pipe. These properties require to be change to suit the actual material.

Amending/Modifying Data and Regenerating Models

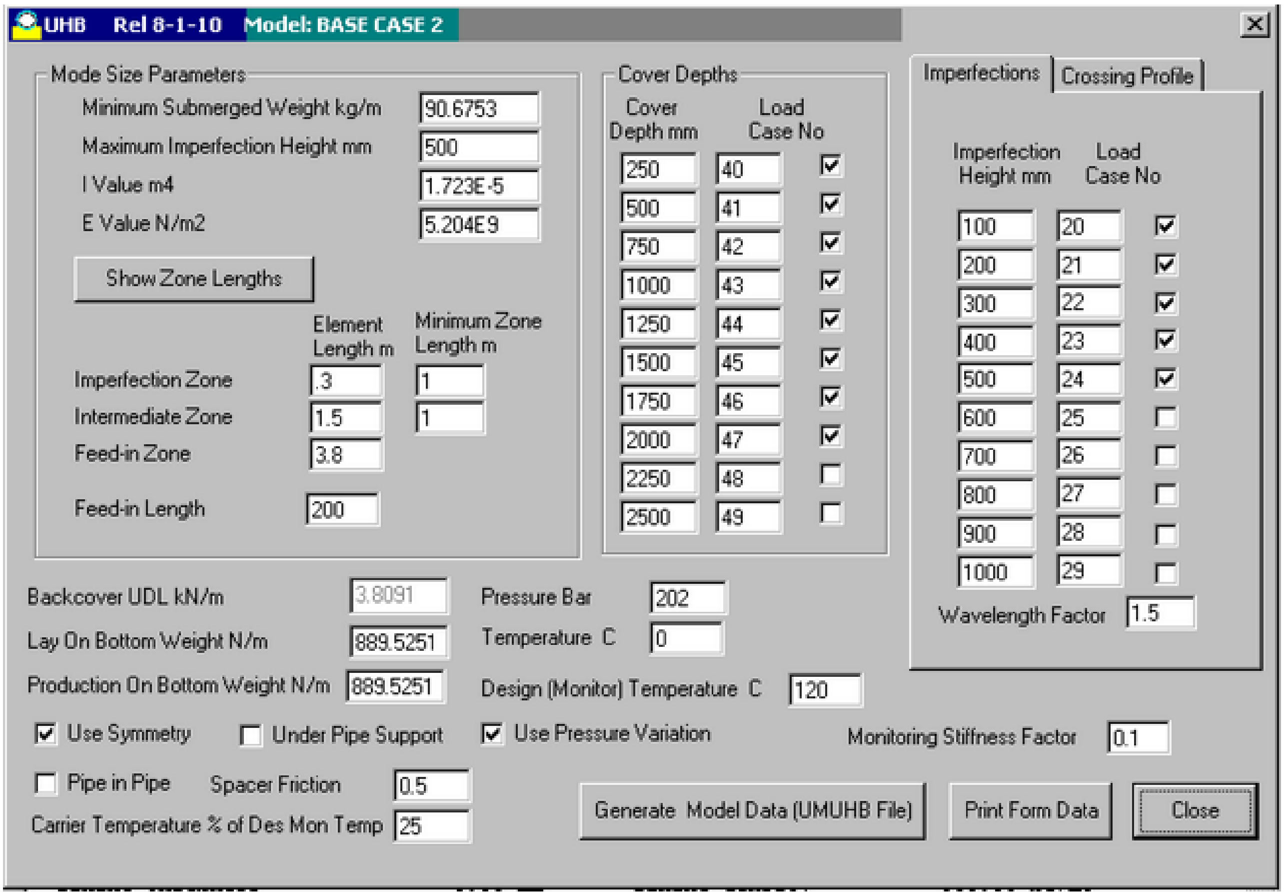
When the UM file is interpreted in Step 8 the model will be 'empty' i.e. no nodes or elements are defined. If this is not the case the interpreted model will be incorrect because the interpreted entities will be appended to the existing model. It therefore essential that if the model is required to be modified, the nodes, the elements and the couples of the existing be deleted (ALL) and the model re-saved before Step 8 is undertaken.

If the pipe properties are require to be used on a different model the existing parameter can be copied by copying and re-naming the **<modelname>.UMPipeProp** and **<modelname>.UMPipeProp1** data files.

-o-

6.1 UHB Definition Data

The following forn is used to define the parameters for the UHB model.



When the **Generate Model Data** button is clicked the form data will be saved to a text file called **<modelname>.UMPipeProp1**

The **Model Size Parameter** data is used to define the length aspects used generate the basic model. The data defined in this panel is used to generate the basic model and the imperfection profiles. It is unlikely that the default values evaluated by the program require to be changed (pipe-in-pipe and flexible pipe will require to be changed). Changing parameters such as the imperfection height, I, E or minimum submerged weight will determine the length of Imperfection Zone establish the size of the model. See 6.0 for note regarding flexible pipe. The default element length are based on the 2.5D(1X), 10D(4X) and 25D(10X) for the different zones. **Note:** The properties in the Model Size Parameters are not used in the model they are only used to generate the length aspects of the model.

The **Show Zone Lengths** button will show the extents of the current model. For some models a warning may recommend reducing the element length in the imperfection zone. The other elements should be changed accordingly using the above length ratios. The Factored Natural Wavelength i.e. the distance from prop to touchdown is given by $L = (\text{Wavelength Factor})(72EI\delta/w)^{0.25}$ The idealised imperfection profile is based on a beam bending cubic deflection profile $y = \delta(x/L)^3 (4 – 3x/L)$.

The default **Feed-in Length** is relatively short as the default model is axially restrained so no realistic feed-in can occur. For the purpose of the prediction of the onset of buckling this lack of feed-in is not a significant restriction. However, if post-buckling deflection is a requirement or expansion analysis is being included then this length would require to be extended beyond the apparent friction anchor location and the axial restraints removed.

The **Backcover UDL** is evaluated by the program for the cover depth defined in the pipe properties form, it cannot be changed. This is initially creates Load Case 12. It is not specifically used by the program because load case 12 is overwritten during batch solution to use the backcover UDL corresponding the cover depth case undergoing solution.

The **Lay On Bottom Weight** is evaluated by the pipe property utility. It is based on flooded pipe. If the pipe is empty this should be changed. This becomes Load Case 10 in the model.

The **Production On Bottom Weight** is evaluated by the pipe property utility. The difference between this and the **Lay On Bottom Weight** is applied in Load Case 13 (applied with the Pressure).

The **Pressure** is used to define the ambient pressure difference ie inner and outer pipe difference. It forms part of Load Case 13.

The **Temperature** defines temperature differential of the production pipe. It forms Load Case 14. The solution times and time curves are set up so that twice this temperature difference is applied to the model. If Pipe-In-Pipe is being analysed the outer pipe can be defined as having a % of the temperature of the inner pipe.

The **Design (Monitor) Temperature** is used the set the temperature at which the force monitoring cases are analysed. This is used in Load Case 15.

The **Use Pressure Variation** option if active will switch the Pressure to Load Case 14 and the Temperature to Load Case 13. This will result in the pressure variation being used to generate the axial force in the pipeline and the temperature being maintained constant The Monitor case also changes from temperature to pressure.

The **Monitoring Stiffness Factor** is used to define the stiffness of the monitoring uplift resistance springs - RC5. So that it a function of cover depth the factor is applied to the downward stiffness magnitude (RC6) . It value may require to be set iteratively set to ensure that displacement are within soil mobilisation limits. The default value for this ratio is based upon the vertical stiffness using the approach described in Section 5.4 for the nominal cover depth defined in the pipe properties form.

The **Use Symmetry** option if active will generate a half model. This option will save time in cases where the model and the loadings are symmetric which is the case for all program generated data. Generating a full model will yield the same results, but just take a bit longer.

The **Under Pipe Support** is used to applied vertical soil support immediately after the pipe imperfection is generated. This would be used in cases where the soil imperfection results in pipe spanning local to the 'prop type' imperfection.

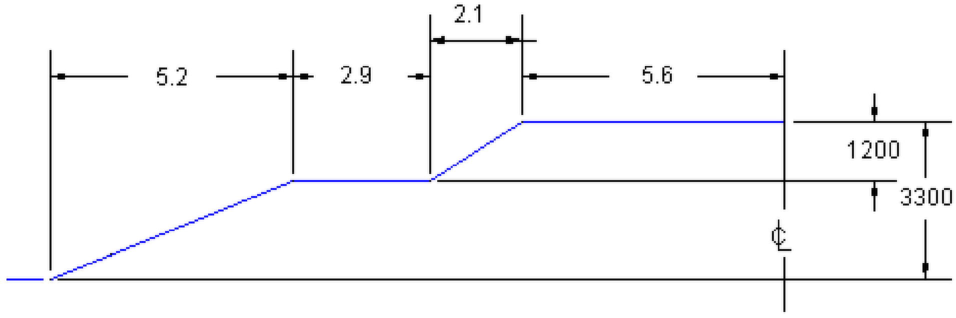
The **Pipe in Pipe** option is used to generate a pipe in pipe model. When this is active the pipe property table must have two entries. The first defines the outer pipe and the second defined the inner pipe. Pipe properties are defined as for single pipe for both but with the following exception. The contents density of the outer is used to define the density of the annular void between the two pipe. If this were air filled, this value would be zero.

The **Spacer Friction** is used to define the coefficient of friction between the inner and outer pipe based on the contact weight between the two.

The **Cover Depth** cases to be considered in the analysis can be defined and selected in the input section. Note that they must be in ascending order.

The **Imperfection** cases to be considered in the analysis can be defined and selected in the input section. Note that they must be in ascending order.

The **Crossing Profile** data can be used to generate a symmetrical crossing profile with the centre of the crossing at the line of symmetry. The convention is that Distance is measured from the centre of the crossing and the lengths are relative. Distance is in m and Height relative to layed pipe in mm. The following indicates the convention used.



The above profile would be defined by:

0	3300
5.6	3300
2.1	1200
2.9	1200
5.2	0

-o-

6.1 UHB Model Description

This section describes the main features of the UHB model. Such an understanding will enable the user if necessary for their application to modify the model.

The model has three basic zones.

- Feed in Zone - Region of coarse Type 8 elements
- Intermediate Zone - Region of intermediate Type 8 elements
- Imperfection Zone - Fine mesh of Type 8 elements

The basic analysis sequence is:

1. With only the vertical and lateral resistance active the on bottom lay weight is applied.
2. The imperfection profile or crossing profile is applied using prescribed displacements applied via gap elements(seabed) imposes an imperfection on straight pipe).
3. The soil cover UDL is applied to the model.
4. The uplift resistance springs are activated
5. The axial resistance springs are activated
6. If the option is active, the backfill downward resistance springs are activated
7. Pressure and the Initial/Operational on bottom weight differential is applied
8. Temperature is applied - The default number of time steps will apply twice the temperature defined in the input form.

Stiffness Properties - Buckling

The soil stiffness properties are evaluated in accordance with the methods described in Section 5.

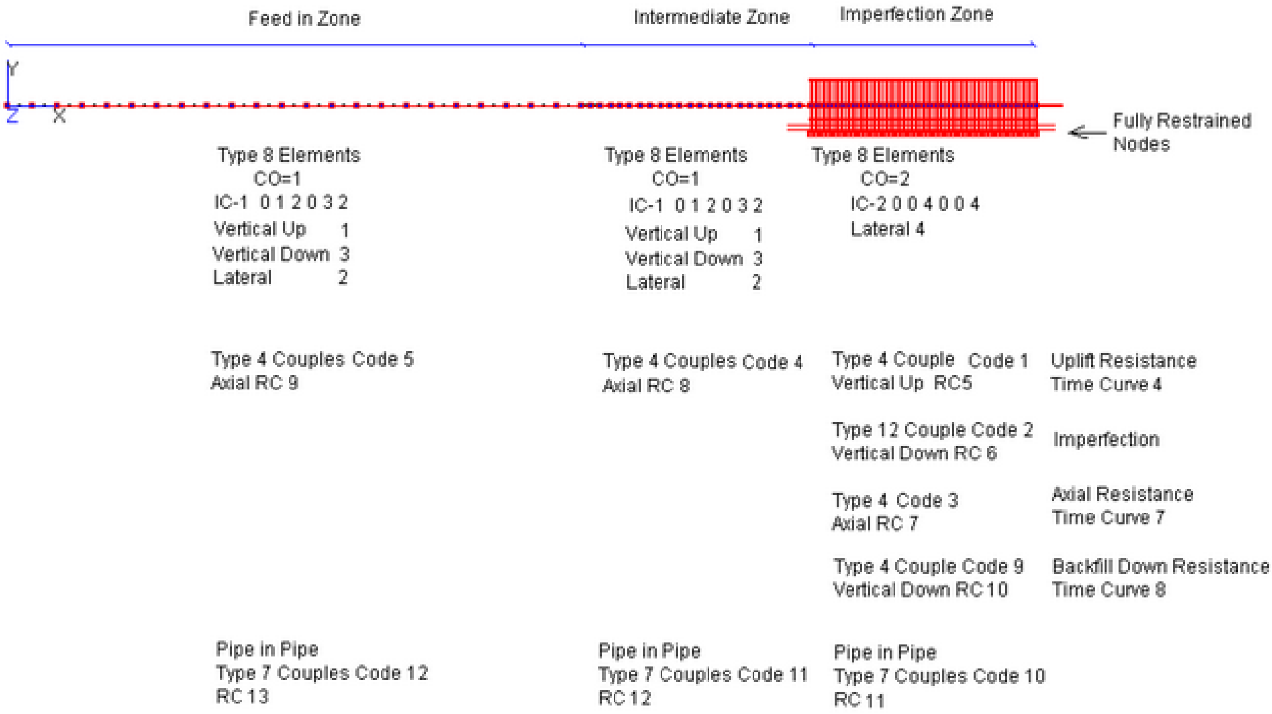
The uplift resistance in the Feed in Zone and the Intermediate Zone have a constant stiffness with no cut off. This is done to prevent buckling occurring outside the imperfection zone (not generally possible but can occur in case where small imperfections and high cover heights are considered).

Stiffness Properties - Monitoring

The stiffness properties for the monitoring case are based on the soil properties corresponding to the nominal cover depth defined in the pipe properties form. The monitoring springs are based on a factor of the stiffness of the vertical down springs. The sensitivity of this value should be checked by inspecting the deflections obtained during uplift (after the imposed imperfection deflection) to ensure that deflections are in a meaningful range ie within the range of the mobilisation distance - see 5.5 .

Model Property Assignment

The following summarises the different element type used in the model.


Feed in Zone
Intermediate Zone
Imperfection Zone
Y
Z
X
Fully Restrained Nodes
Type 8 Elements
CO=1
IC-1 0 1 2 0 3 2
Vertical Up 1
Vertical Down 3
Lateral 2
Type 8 Elements
CO=1
IC-1 0 1 2 0 3 2
Vertical Up 1
Vertical Down 3
Lateral 2
Type 8 Elements
CO=2
IC-2 0 0 4 0 0 4
Lateral 4
Type 4 Couples Code 5
Axial RC 9
Type 4 Couples Code 4
Axial RC 8
Type 4 Couple Code 1
Vertical Up RC5
Uplift Resistance
Time Curve 4
Type 12 Couple Code 2
Vertical Down RC 6
Imperfection
Type 4 Code 3
Axial RC 7
Axial Resistance
Time Curve 7
Type 4 Couple Code 9
Vertical Down RC 10
Backfill Down Resistance
Time Curve 8
Pipe in Pipe
Type 7 Couples Code 12
RC 13
Pipe in Pipe
Type 7 Couples Code 11
RC 12
Pipe in Pipe
Type 7 Couples Code 10
RC 11

-o-

6.2 Load Cases & Combinations

Buckling Curve Generation

The following shows the basic load cases and the combination types which are used to generate the buckling curves.

Combination 101

Buckle300mm Imperfection

10	1	Submerged Lay Weight
20	2	100mm Imperfection
12	3	Backcover Resistance
13	5	Pressure 200 b & Contents Diff
14	6	Temperature 200 Amb 4

This is a DyNoFlex solution so all load factors refer to time curves.

The CFACT commands used to activate the soil springs are all contained in L10.

For each imperfection height a different combination is used. The only difference between these combinations is the imperfection load case used eg L20 in the case shown above.

The different cover heights are accounted for in the batch file. For each cover height the same combinations are used but the Backcover Resistance load case L12 is overwritten and the results overwritten. For each height case run the result data used to create the buckling curves are saved in the <model>.~fhplot(n) file which are copied as the solution progresses. This can be seen in the following extract from a typical buckling curve batch file.

```
modmerge m/mdl/UMUHB_250/  ------   The UMUHB_250 file contains the uplift resistance curves
corresponding the 250mm soil cover.
winfram i
MFCOPY "L40" "L12" -------  Copies overburned UDL for 250mm cover
BAND 1
NLPARAM 101
LOADA C101/0/0/1/101/
DYNFLEX_SS1
MFCOPY "~fhplot101" "~fhplot1"  ----- Copies plot file uses for element force plots
.
.
NLPARAM 105
LOADA C105/0/0/1/105/
DYNFLEX_SS1
MFCOPY "~fhplot105" "~fhplot5"
modmerge m/mdl/UMUHB_500/
winfram i
MFCOPY "L41" "L12"
BAND 1
NLPARAM 101
LOADA C101/0/0/1/101/
DYNFLEX_SS1
MFCOPY "~fhplot101" "~fhplot6"
.
.
NLPARAM 105
LOADA C105/0/0/1/105/
DYNFLEX_SS1
MFCOPY "~fhplot105" "~fhplot10"
modmerge m/mdl/UMUHB_750/
winfram i
MFCOPY "L42" "L12"
BAND 1
NLPARAM 101
LOADA C101/0/0/1/101/
DYNFLEX_SS1
MFCOPY "~fhplot101" "~fhplot11"
NLPARAM 102
```

Downward Force Monitoring

The following shows the basic load cases and the combination type which are used to generate the cover weight requirements.

Combination 201

Monitor100mm Imperfection

10 1
20 2
13 5
15 6

For each imperfection height a different combination is used. The only difference between these combinations is the imperfection load case used. The following shows a typical batch file.

```
modmerge m/mdl/UMUHB_Monitor/
winfram i
BAND 1
NLPARAM 201
LOADA C201/0/0/1/201/
DYNFLEX
NLPARAM 202
LOADA C202/0/0/1/202/
DYNFLEX
NLPARAM 203
LOADA C203/0/0/1/203/
DYNFLEX
NLPARAM 204
LOADA C204/0/0/1/204/
DYNFLEX
NLPARAM 205
LOADA C205/0/0/1/205/
DYNFLEX
```

Running the Batch File

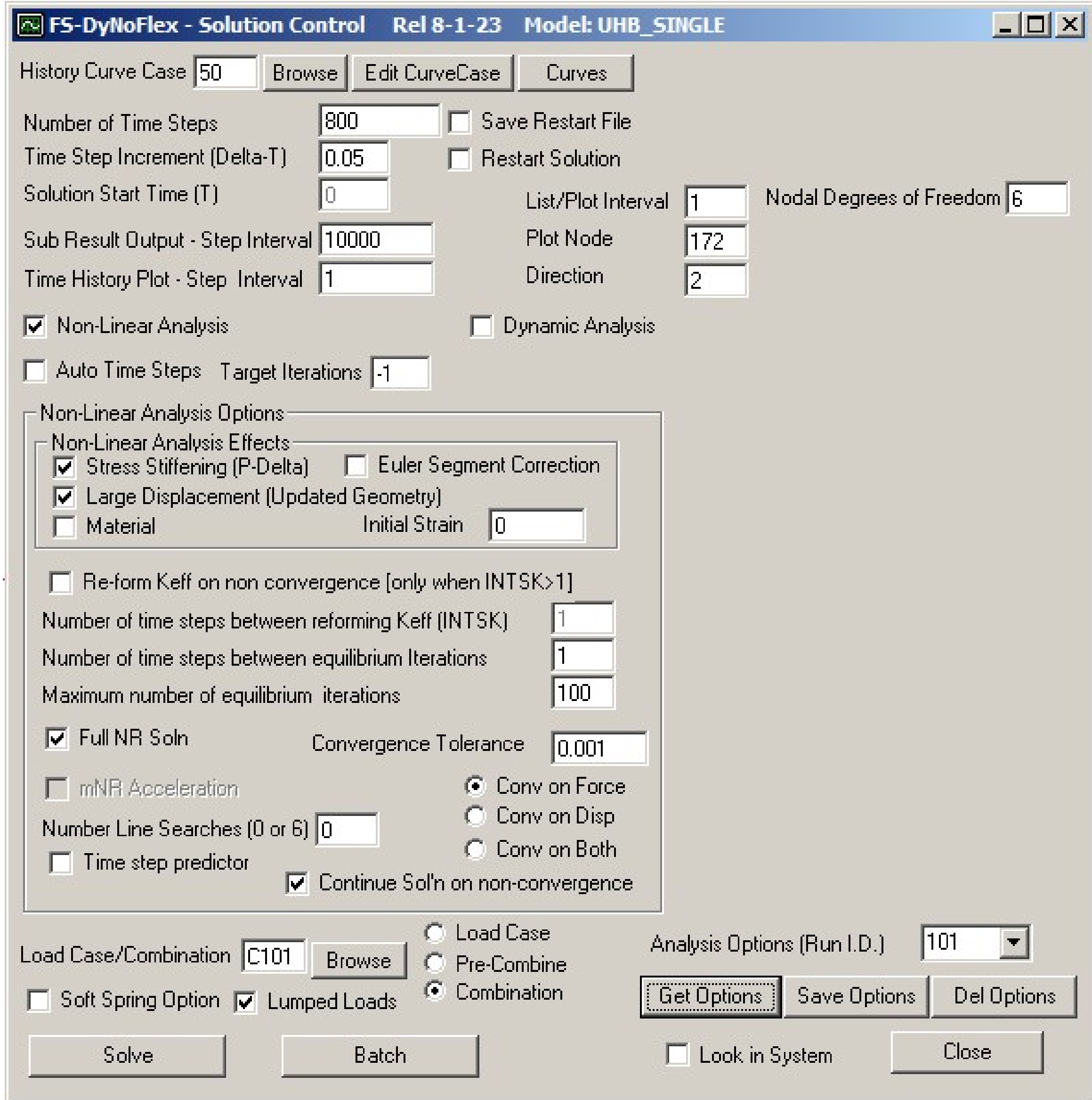
Usually the batch file will run and the all solution runs will be completed. Occasionally with buckling analysis the solution may cease at the point of the first instability and the post buckling behaviour will not be possible using the current DyNoFlex setting. In this particular type of analysis it is not important to trace post buckling behaviour as the object of the analysis is only to predict it. Even if the solution fails the buckling load is recorded and meaningful result will be obtained. On some models the batch file will stop because DyNoFlex has encountered a non-trapped error. In such cases close the DyNoFlex Windows and allow the batch to continue.

-o-

6.3 Time Curves & DyNoFlex Options

The following time curves are used to apply the loading and activate the soil springs.

```
-----------On Bottom as layed Weight----------------------
tcurve,1,3,1
0,0
5,1
1000,1
------------------Imperfection-----------------------------------------
tcurve,2,4,1
0,0
5,0
10,1
1000,1
------------------Soil Cover---------------------------------------
tcurve,3,4,1
0,0
10,0
15,1
1000,1
------------------CFACT CURVE Uplift------------------------------
tcurve,4,4,1
0,0
15,0
15.0001,1
1000,1
------------------Pressure----------------------------------------
tcurve,5,4,1
0,0
15,0
20,1
1000,1
------------------Temeprature-------------------------------------
tcurve,6,4,1
0,0
20,0
30,1
100,8
------------------CFACT CURVE Axial------------------------------
tcurve,7,4,1
0,0
10,0
10.0001,1
1000,1
------------------CFACT CURVE Imperfection Vertical Support--------
tcurve,8,4,1
0,0
10,0
10.0001,1
1000,1
```


FS-DyNoFlex - Solution Control Rel 8-1-23 Model: UHB_SINGLE
History Curve Case 50
Browse
Edit CurveCase
Curves
Number of Time Steps 800
Save Restart File
Time Step Increment (Delta-T) 0.05
Restart Solution
Solution Start Time (T) 0
List/Plot Interval 1
Nodal Degrees of Freedom 6
Sub Result Output - Step Interval 10000
Plot Node 172
Time History Plot - Step Interval 1
Direction 2
Non-Linear Analysis
Dynamic Analysis
Auto Time Steps
Target Iterations -1
Non-Linear Analysis Options
Non-Linear Analysis Effects
Stress Stiffening (P-Delta)
Euler Segment Correction
Large Displacement (Updated Geometry)
Material
Initial Strain 0
Re-form Keff on non convergence [only when INTSK>1]
Number of time steps between reforming Keff (INTSK) 1
Number of time steps between equilibrium Iterations 1
Maximum number of equilibrium iterations 100
Full NR Soln
Convergence Tolerance 0.001
mNR Acceleration
Conv on Force
Conv on Disp
Number Line Searches (0 or 6) 0
Conv on Both
Time step predictor
Continue Sol'n on non-convergence
Load Case/Combination C101
Browse
Load Case
Pre-Combine
Combination
Analysis Options (Run I.D.) 101
Soft Spring Option
Lumped Loads
Get Options
Save Options
Del Options
Solve
Batch
Look in System
Close

-o-

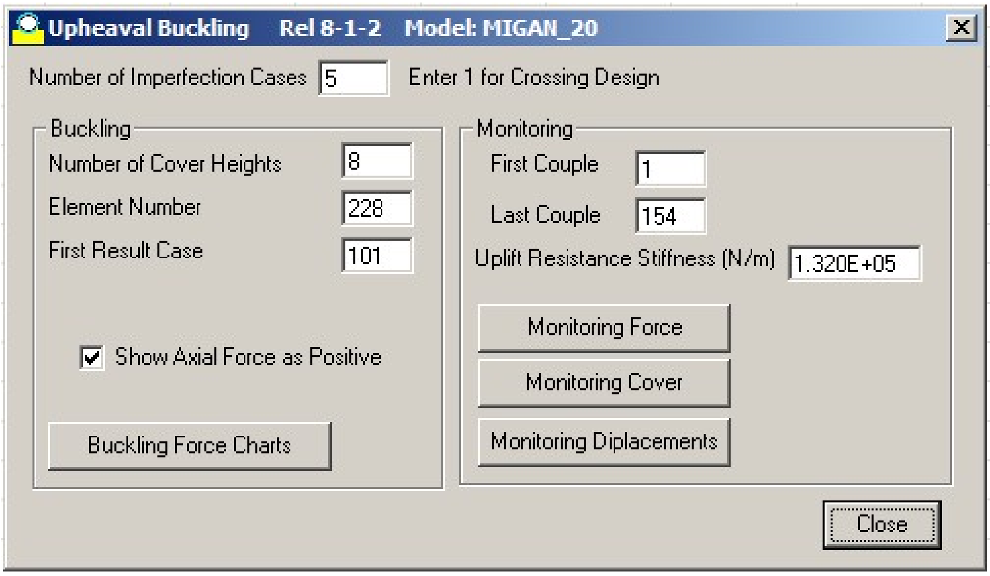
6.4 Extracting Results from a UHB Model

This section described how the results from a UHB solution are accessed.

6.4.1 Special Utility

The objective of the UHB analysis is to establish the relationship between the axial force and minimum cover height to prevent buckling for a given imperfection.

A special utility program is used to extract this data from the results. This is started from the Windows start menu (FS2000/Pipe Utilities/UHB Results).
This will create and load an Excel spreadsheet containing either the buckling curves or the monitoring data. It also create a text file containing the same data -.



To access the buckling data the **<model>.BRM** should have been run.

To access the monitoring data the **<model>.BRM_Mon** should have been run.

The data should not require to be changed unless the model has been modified from that generated from the UHB model generator.

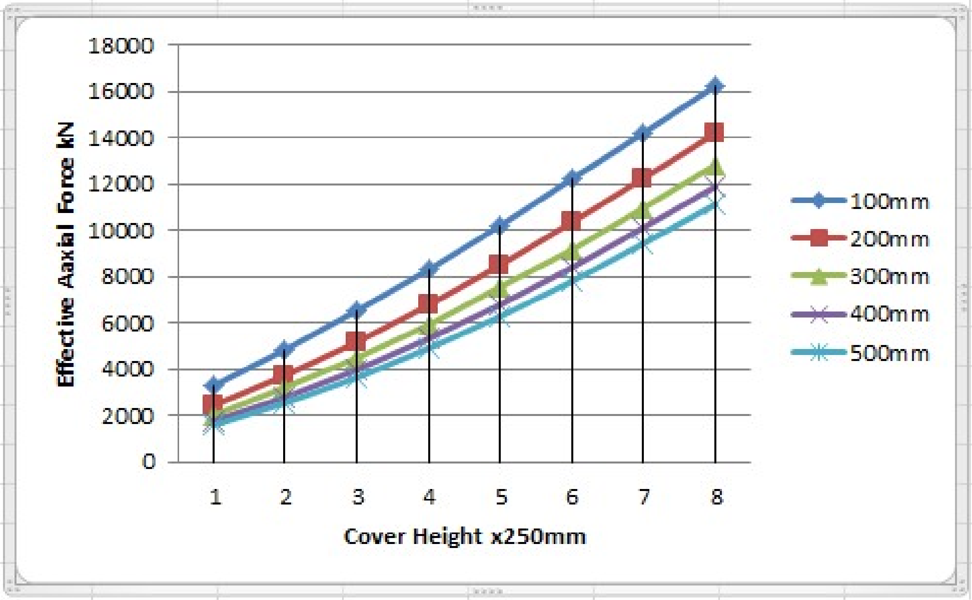
The **Element Number** is the element whose axial force is to be monitored. The default is located at the maximum prop height.

The **First Couple** located at the start of the imperfection zone and the **Last Couple** is located at the maximum prop height. These couples are the vertical monitoring springs.

The **Uplift Resistance Stiffness** is the stiffness of the monitoring springs. The default is the stiffness (RC5) generated by the UHB utility when the model data was created the and is governed by the Monitoring Stiffness Factor. It is only used to evaluate the spring displacements. If RC5 in the model is changed (sensitivity studies) then this value must reflect this value otherwise the displacement will be incorrectly evaluated.

When the **Buckling Force Chart** button is clicked an Excel sheet will be created containing the axial buckling forces (kN) in a table. The table columns represent the number of cover heights. The table rows represent the number of imperfection cases.

	A	B	C	D	E	F	G	H
1	3288.221	4815.138	6504.539	8321.193	10220.84	12202.79	14185.68	16209.61
2	2412.39	3729.444	5214.708	6824.887	8518.29	10335.43	12235.25	14176.69
3	2027.972	3182.208	4501.888	5947.041	7516.415	9209.538	10985.56	12844.36
4	1770.962	2800.242	3995.897	5316.565	6802.079	8371.865	10104.78	11881.45
5	1594.104	2539.829	3652.24	4889.972	6252.376	7779.004	9389.975	11123.88



If Excel is not present of fails to run, the data will be echoed in a text file called **<model>.~UHBuck**

When the **Monitoring Force** button is clicked an Excel sheet will be created containing the forces (kN) in the monitoring springs along the imperfection zone. The table columns represent the number of imperfection cases. The table rows represent the spring location (1 is the start of the imperfection zone). The data will be echoed in a text called **<model>.~UHBMonF**

When the **Monitoring Cover** button is clicked an Excel sheet will be created containing the required cover (m) to provide the buckling resistance along the imperfection zone. This value is calculated (quadratic) using the soil stiffness model described in Section 5.4. The table columns represent the number of imperfection cases. The table rows represent the spring location (1 is the start of the imperfection zone). The data will be echoed in a text called **<model>.~UHBMonC**

When the **Monitoring Displacements** button is clicked an Excel sheet will be created containing the displacements (mm) in the monitoring springs along the imperfection zone. The table columns represent the number of imperfection cases. The table rows represent the spring location (1 is the start of the imperfection zone). The data will be echoed in a text called **<model>.~UHBMonD**

6.4.2 Using FS2000 - Results TASK

The batch file runs the buckling solution in a series with each cover height being common and the imperfection height varying. The results for each cover height are overwritten but the files that contains the time solution data is copied. This enables all solution time history element force data to be plotted.

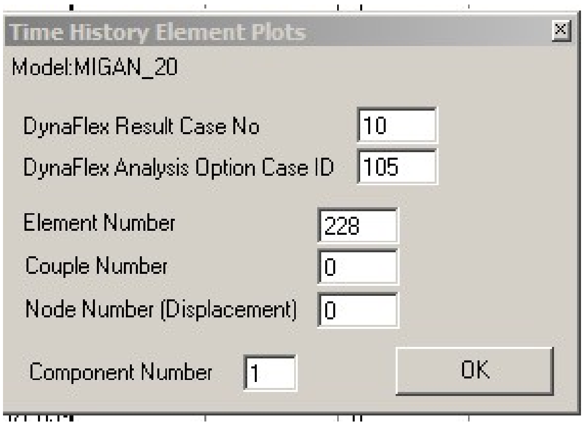
The results from the UHB analysis can be post-processed like any other result case and the results interrogated in the Results TAST. If 5 imperfection case are used result cases 101 - 105 will exist for the buckling curves and 201 - 205 for the monitoring cases. The 101 - 105 results will be the cases for last cover height. If other previous cover heights are required to be investigated simply run that section of the batch to re-create the results.

A typical run with 5 imperfection cases and 8 cover heights will produce the 40 force plot files. The numbering sequence is as indicated the table below.

Analysis/Options Case	Re-numbered history plot
101	1
102	2
↓	
105	5
101	6
↓	
105	10
↓	
105	40

When loading these into the FS-Graph Time History Plot routine the DyNoFles Result Case Number should be the Re-numbered case and the DyNoFlex Option Case the Option Case No.

The following would plot the axial force in E225 for the 5th imperfection case and 2nd cover height.



The monitoring case only using the 5 imperfections cases and are these not overwritten, therefore these results can be post-processed and viewed as per any other DyNoFlex result case.

-o-

7.0 Expansion Analysis - Model Generator

The Expansion Model Generator is a utility that will generate a complete pipeline expansion model including load cases. If the time history option is active the utility will also create a the FS-DyNoFlex options file and time curve to undertake the thermal cycling analysis.

The input from becomes visible when the **Expansion Model** button is clicked

The main features of the utility are:

- All pipe properties, seabed stiffness and loadings are generated from pipe data input form
- Single pipe and Pipe-in-Pipe models can be generated
- Pipe element temperature data is generated from simple user defined temperature profiles
- Time dependent profile definition can be used to investigate pipe walking

The following basic procedure should be followed to create the model and complete the analysis.

1. Open a new model. Do not generate any data.
2. Create the temperature profile <model>.UTP(n) files.
3. Start the Pipe Properties Utility
4. Use the Pipe Properties to define the pipe and foundation properties and save the table. The **Surface Pipe** option must be active for pipe walking analysis.
5. Click the **Expansion Model** button to make the Expansion data form visible
6. Enter the expansion design parameters appropriate to the pipeline configuration.
7. Click the **Generate Model Data (UMEXP File)** button.
8. Return to the FS2000 GUI
9. In the Model Definition TASK use the Interpret File command to interpret the <Model>.UMEXP file.
10. Save the model
11. Go to the Primary TASK and Archive the model.
12. Open the model from its Archive format.

If no temperature profile cases are defined then only two load cases will be created, the on bottom weight case (L10) and a constant 200C temperature case (L11).

The resulting model can be run for simple expansion analysis using the 3D NL/Pile Analysis Option or the FS-DyNoFlex Option. However, only the latter can used for pipe walking analysis since the solution requires the use of time curves to apply the sequential loadings.

Amending/Modifying Data and Regenerating Models

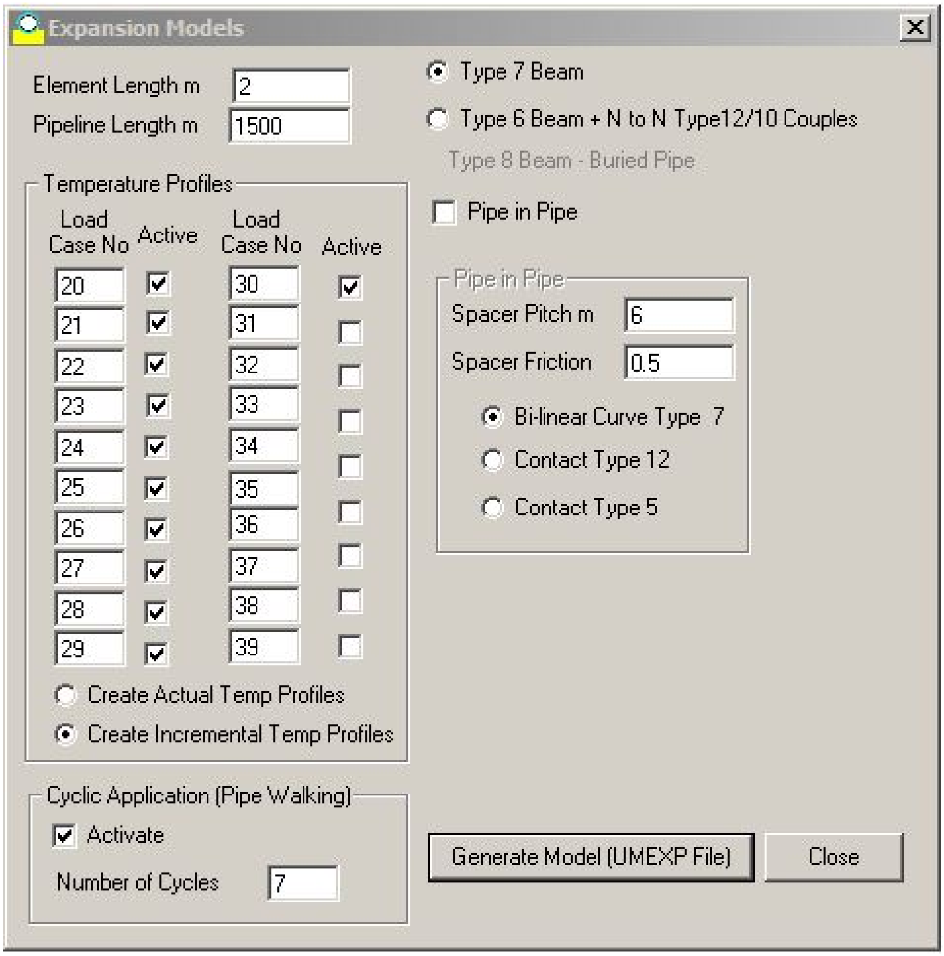
When the UM file is interpreted in Step 9 the model will be 'empty' i.e. no nodes or elements are defined. If this is not the case the interpreted model will be incorrect because the interpreted entities will be appended to the existing model. It therefore essential that if the model is required to be modified, the nodes, the elements and the couples of the existing be deleted (ALL) and the model re-saved before Step 9 is undertaken.

If the pipe properties are require to be used on a different model the existing parameter can be copied by copying and re-naming the **<modelname>.UMPipeProp** and **<modelname>.UMPipeProp2** data files.

-o-

7.1 Expansion Definition Data

The following form is used to define the parameters for the Expansion model. It is used both for single case expansion and sequential case expansion (pipe walking analysis). The option are save when the model data is generated.



When the **Generate Model** button is clicked the form data will be saved to a text file called **<modelname>.UMPipeProp2**

The **Element Length** set length of the element. This is constant over the whole model. If pip-in-pipe is being generated the inner pipe will also have same length. Void making this too small otherwise the model will be unnecessarily large.

The **Pipeline Length** define the total length of the pipeline.

For surface pipelines the pipe/seabed interaction can be modeled with Type 7 elements or Type 6 element + Type 12 couple elements. The latter option is should be used if the seabed reaction requires to be plotted. For fully buried pipelines Type 8 elements will be used.

The **Pipe in Pipe** option if active the utility will generate a pipe in pipe model using the parameters defined in the Pipe-in-Pipe options panel which will become active. When this is active the pipe property table must have two entries. The first defines the outer pipe and the second defined the inner pipe. Pipe properties are defined as for single pipe for both but with the following exception. The contents density of the outer is used to define the density of the annular void between the two pipe. If this were air filled, this value would be zero.

The **Temperature Profiles** defined the temperature cases to be created. A <model>.UTP(n), where n is the load case number must exist for a case to be activated. Note that temperature profile can be plotted within the Load Definition TASK using the TP button (Pipe Tool Bar).

The temperature profile file is a simple text file which defined the profile along the pipeline, the same file is used by FS2000's (Load Defintion TASK) **ETPR** command . Only the key points of the profile are require, the routine will interpolate the profile and define the element temperature profile (note that the model element numbering must start at label 1).The file format for the **<model>.UTP(n)** profile file is:

Number of points
Distance1, Temperature1
….. , ……
DistanceN,TemperatureN

A simple linear profile would be:
2
0, 200
1500,150

If the **Create Actual Temp Profiles** option is active the load case generated will be the actual profiles defined in the UTP files. If the **Create Incremental Temp Profiles** option is active the load case generated will be the difference between current temperature profile of that of the preceding case. For pipe walking analysis the **Create Incremental Temp Profiles** option must be active.

Cyclic Application (Pipe Walking)

If the **Sequential Time History** option is active then a Loads Case Combination (LC100) will be created which is associated with the time curves defined in load case L50 which is also created. A FS-DyNoFlex options file (100) will also be generated which can be used to run the loading time history analysis. The load cases are applied in the order of the list shown. If the **Sequential Time History** option is not active only the temperature profile load cases will be created ie for individual solution using the Standard Non-Linear solver.

The **Number of Cycles** is used to define the number of times the time temperature profiles case are required to be repeated. This number sets the maximum time duration for the FS-DyNoFlex solution and the time curve duration limits.

-o-

8.0 Reeling Analysis - Model Generator

The Reeling Model Generator is a utility that will generate a pipeline reeling model. This generator is used in conjunction with a template model (ReelTemp.MOD) to create a full analysis model. The generator creates the model and the template model provides the load case combinations, FS-DyNoFlex option settings and the batch files.

The main features of the utility are:

- All pipe geometry and load cases created
- Nominal non-linear properties are generated
- Single pipe and Pipe-in-Pipe models can be generated
- Plastic stress and strains are evaluated

The following basic procedure should be followed to create the model and complete the analysis.

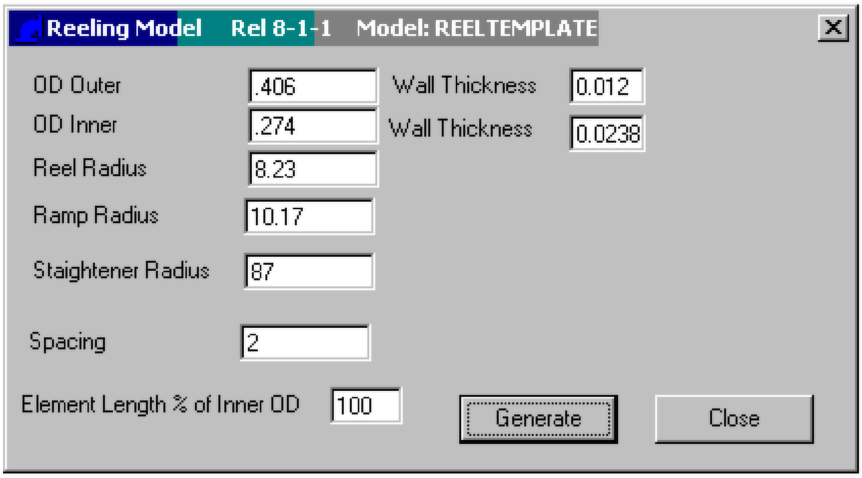
1. Open a template model file (**ReelTemp.MOD**).
2. Start the Reeling Model Generator
3. Use the Pipe Properties to define the pipe properties and associated reeling radii
4. Click the **Generate** button.
5. Return to the FS2000 GUI
6. In the Model Definition TASK use the Interpret File command to interpret the <Model>.UMREEL file.
7. Save the model
8. Run the batch file to obtain the results

Note that to generate a straight pipe will require an iterative approach and it is unlikely that a perfectly straight pipe will be obtained. When changing the straightener radius there is no reason to re-interpret the model as this will only effect the load cases generated.

-o-

8.1 Reeling Model Definition Data

The following form is used to define the parameters for the Reelin Model model.



The generator will only generated pipe-in-pipe model so if only a single pipe is required, the inner pipe (Property Code 2) will have to be deleted in FS2000. The connection Couples (Property Code 1) will also have to be deleted.

Note: All radii are mean pipe radii.

Pipe strain = D / (2R + D) where D = pipe od and R = reel radius

-o-