FS2000

Pile Analsysis and Design

Advanced Structural Analysis
for Windows

Email:support@aes-uk.com

Table of Contents

1.0 Introduction

FS-Pile is a program module that interfaces with FS2000 for the non-linear analysis of single piles under axial and lateral loading. The program evaluates load capacities and the associated stiffness characteristics (PY, TZ & QZ response curves). The module can undertake pile head load response analysis without having to use FS2000's solvers. It can, and is more commonly used to generate pile non-linear pile elements for inclusion into FS2000 structural models.

The pile capacities are be evaluated using the different methods presented in the API RP2A 21st Edition (including the 2008 supplement) and the ICP method (MTB publication 96/103) . Capacities can also be evaluated using defined stiffness curves e.g. P-Y curves.

Any type of soil cohesive\non-cohesive profile can be defined.

API methods are implemented for the generation of the stiffness characteristcs of the pile but optional variations are available so as to be consistent with other industry practices.

Pile driving analysis can also be undertaken using Smith's Wave Equation.

The main operational features are:

- PY, TZ & QZ Curves generated from soil design strength profiles using API methodology
- User defined PY, TZ & QZ option for input
- Full graphical output of P-Y data etc.
- Axial capacities (inc. FoS) based on skin friction and end bearing
- Graphical plots of pile data with depth i.e. Axial capacity skin friction etc.
- Pile head forces or displacements may be analysed
- Graphical plots of pile head stiffness characteristics
- Generation of non-linear pile elements for full interaction with 3-D structural models
- Automatic generation of linear equivalent pile and springs
- Pile driving analysis using Smith's Wave Equation

Note that to analysis 3-D pile/structure interaction requires the use of the optional FS-3D Non-linear analysis module.

-o-

2.0 Basic Program Operation - Overview

The following outlines a basic pile analysis procedure for single pile analysis on new model where there is no existing model data. The procedure is basically the same if piles are to be added to an existing model (see below).

- Create a new model in FS2000 - Do not enter any model data.
- Start FS-Pile from the Windows Start menu (FS2000 program group).
- Data relating to the pile is interactively input using the main FS-Pile window form. The pile data must then be saved as a pile model data using the **Save** command in the File menu. By default the FS2000 model name will be shown - use this. If soil profiles or user defined PY curves are to be used these are required to be created by the user (see Sect 5 for file formats and file extensions) using the same file name as the FS2000 model. A soil data file editor is provided (Soil Data Editor command:File menu) that enables data to be conveniently entered in a data grid.
- Use **Evaluate Properties/Capacities** command in the Analysis menu to create the P-Y data, non-linear spring data and to evaluate the axial capacities of the pile. This data created during this process may be viewed in tabular form using the View Capacity Data command in the view menu. Plots of this data can also be viewed using the commands of the plot menu. Use this option to design size the pile.
- Use **Generate Non-Linear Pile Model** command in the Analysis menu to generate the pile structural data for the FS2000 model. This command creates a <name>.UM1 model definition file. To input the pile structural data to the FS2000 model the UM1 must be interpreted (Interpret File command:File menu) from within the Model Definition Task of FS2000. The model must then be saved using the same model name and directory as that used to save the pile model data. The nonlinear soil springs are associated with the generated pile elements.
- The **Run Pile Stiffness Analysis** command in the analysis menu can now be used to create or run loads cases for 2-D pile analysis. When this command is activated a form for the input of pile head forces or displacements becomes visible. These defined loads can be saved as load cases and then run directly from the window form. Pile head load-displacement plots of the results can be viewed using the Plot menu. If the results are post-processed in FS2000 these results can be viewed in FS2000 i.e. moment plots and stress output created.

  Note: Load Cases can be added to the pile and saved within the Load Definition Task of FS2000 just like a conventional model BUT the pile analysis can only by run from FS-Pile or the FS2000's non-linear solvers.

- 3-D Pile models are solved using the 3-D Non-Linear or DyNoFlex (Analysis TASK/Solution menu). Note: When running 3-D pile solutions an additional pile tip torsional restraint may be required - pile springs only provide lateral and axial restraint.
- The Equivalent Linear Pile command in the analysis menu is used establish an equivalent linear beam (+ pile head lateral springs) for a defined pile head load condition. This operation uses the Batch Control Module of FS2000 since multiple load cases are required to be run (done automatically).

Adding Pile to Existing Models (Extended Pile Models)

The above procedure assumes a new FS2000 structural model is being used (step 1 above). Pile elements can be added to any existing FS2000 model. The only difference in the procedure is that when the pile segment definition data is being generated (UM1 file) the start label for the nodes and elements of the pile must different from those in the existing FS2000 structural model. This is a requirement to prevent the existing model entities being overwritten when the UM1 file is interpreted.

Similarly additional nodes and elements can be added to any pile model in FS2000. When additional nodes or elements are added to the pile model the existing element labels of the pile elements must remain unchanged as they are linked to the non-linear springs generated by FS-Pile.

A procedure for creating and running extended pile models is given in Sect 3.7.

Solution of Models with Piles

The standard linear solution module of FS2000 cannot be used with pile models since it does not recognise the non-linear pile springs.

The only solvers that can be used are:

- The **Run Pile Stiffness Analysis** command in FS-Pile used for 2-D pile models
- The **3-D Non-Linear**, the **2-D Pile Analysis** or **DyNoFlex**

Normally only two pile related files are generated and used in the solution. The UPN file which associates Type 6 element with soil spring data and the UNP file which contains the soil spring data. There is an option to generate use Type 7 non-linear springs and this approach does not use these files and the pile elements are not required to be Type 6 elements, they can be linear.

Additional restraints can be applied to pile models so that they can analysed with the linear solver. This is sometimes useful for debug purposes if a model is not running for correctly for reasons other than pile supports i.e. fully restrain the pile nodes at a point just below the surface It theses additional restraints are removed the model can then be again be run as a pile model.

Sensitivity Studies

If the length of the pile is changed in FS-Pile it is necessary to re-generate the pile model in FS2000. This is a requirement since the number of nodes and elements in the pile segment will change as the length changes. In such cases the existing pile nodes and elements should be deleted prior to re-interpreting the UM file (not required if the existing data is always overwritten).

The reason this has to be done is that the pile springs are associated with specific elements. The elements are generated to match the soil data therefore if the soil stratification changes or the pile length changes the element's length has to change. If the soil shear strength or the friction angle are the only changes then it is not necessary to re-interpret the UM files into the model because the **Evaluate Properties/Capacities** will update the spring related UNP file**.**

If the wall of the pile is changed the property code in FS2000 must be modified accordingly or alternatively the pile model must be re-generated. Only OD changes require pile spring property (UNP file) re-evaluation.

-o-

2.1 Units

The program only operates using FS2000 S.I. units. If can however be used to generate non-linear piles for US-Unit models.

The following restrictions/requirements relate to **US-Unit** models:

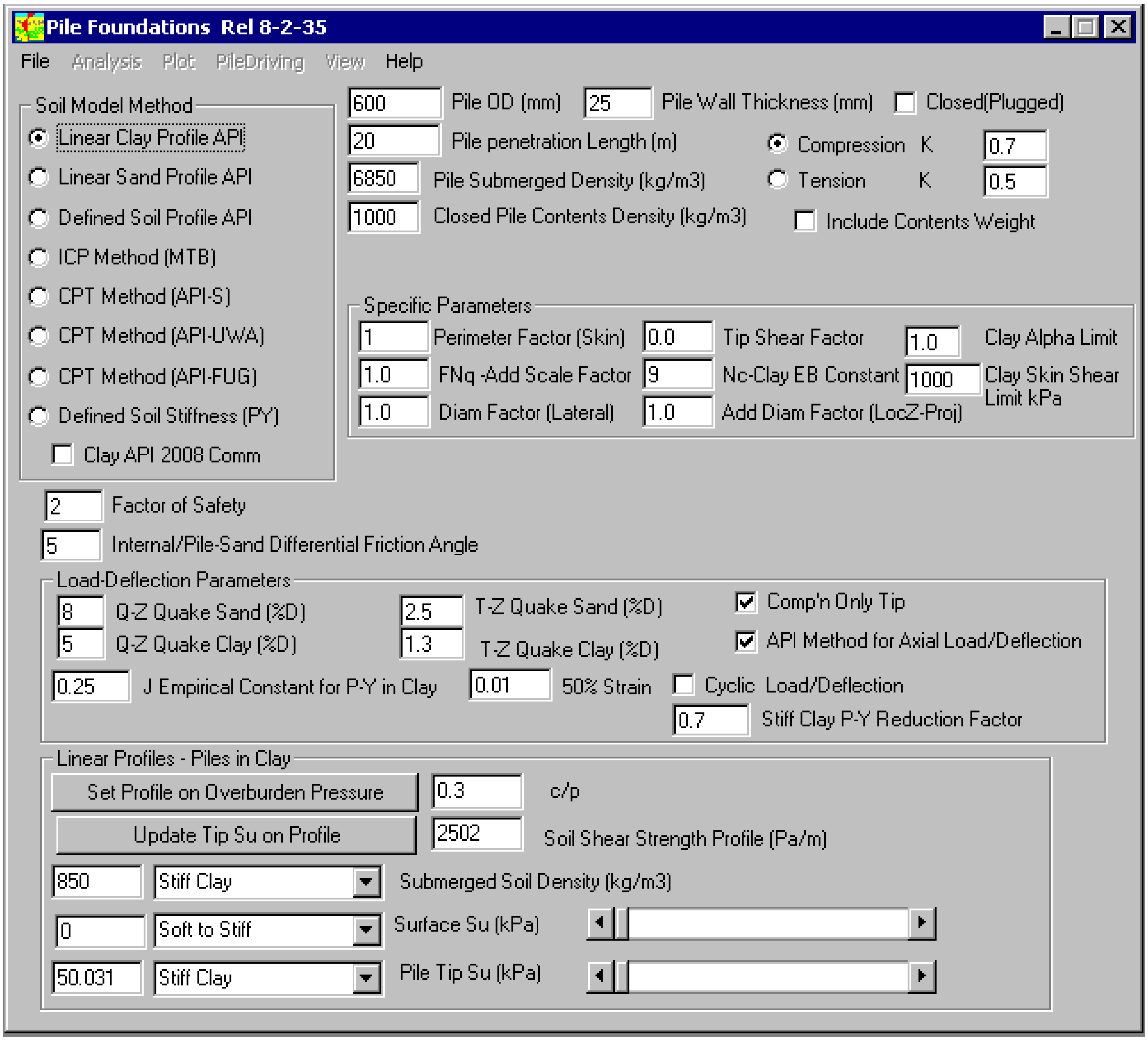
- All soil data must be defined in the derived S.I. unit as described in this Help file.
- All evaluated capacity data (including P-Y curves) will be in S.I derived units.
- Generated pile and their associated solution stiffness data will be compatible with US-Unit models i.e. Ins and Lbs used (<model>.UNP file),
- When generating non-linear piles their coordinates must be defined using the model length unit i.e. ins
- Equivalent linear piles can not be evaluated in US-Unit models,

-o-

3.0 Program Operation

Clicking the Pile Analysis icon in the FS2000 Window group starts the program. When this is done the main FS-Pile form will appear.

The menu commands will not become active until the pile data is saved or an existing pile data file is opened.

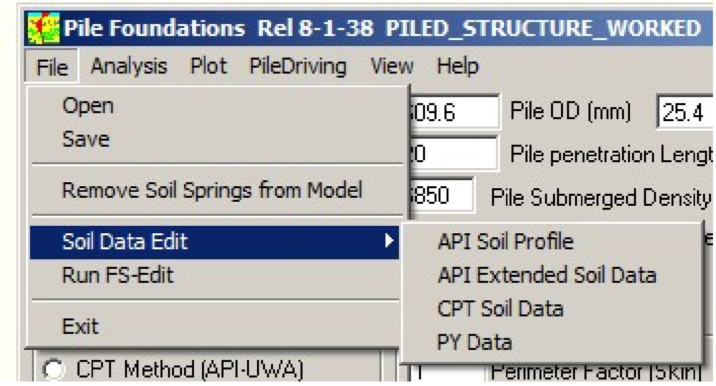


-o-

3.1 Menus

The operation of the FS-Pile is undertaken using command menus. The main menus are given below. Note that the menus are inactive until a pile data file is opened or pile data is saved to a file. It is recommended that the pile data file is ALWAYS the current FS2000 model. This will appear as the default.

File



Open — Used to open existing pile data files. Always use FS2000 default.

Save — Saves current data to a pile data file. Always save either before or after using the Analysis menu commands to generate pile properties.

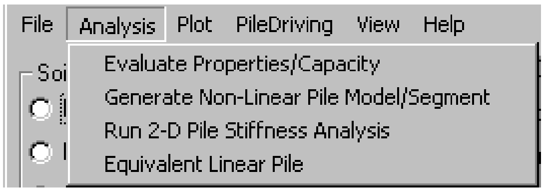
Remove Soil Springs — This deletes the UNP and UPN files. Enables previously assigned pile element (same label) to be used as standard beam elements.

Soil Data Edit — Create and edit soil data in a grid editor.

Run FS-Editor — Runs FS2000 text editor.

Exit — Exits program

Analysis - *This menu is described in more detail Section 3.6*



Evaluate Properties/Capacities — Evaluates pile properties and load capacities. This includes the stiffness pile stiffness properties i.e.the P-Y data etc. and the element soil spring file (UNP).

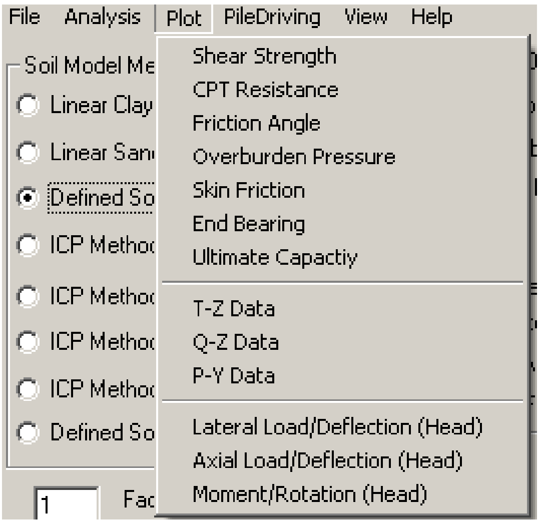
Generate Non-Linear Pile Model — Evaluations as previous command but also creates an FS2000 model geometry definition file (UM file) and the element/soil spring association file (UPN).

Run Pile Stiffness Analysis — Creates and runs 2-D load cases (pile head load conditions). To run an analysis, the pile model must have first been saved in FS2000.

Equivalent Linear Pile Element — Evaluates the beam and spring properties for an equivalent linear beam based on specific loading conditions.

Plot - *To activate the following property plots the* ***Evaluate Properties/Capacities*** *option in the*

Analysis *menu must have been run.*



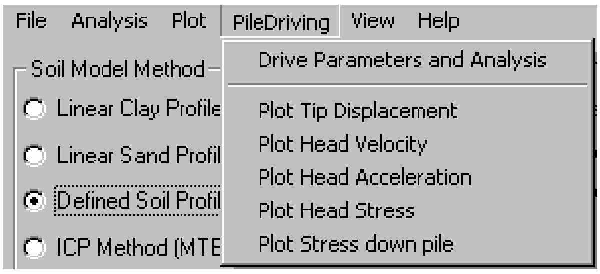
To activate the following pile stiffness shown below plots the ***Run 2-D Pile Stiffness Analysis*** *option in the* ***Analysis*** *menu must have run the subject load case. This plot facility is only for isolated pile models and not extended pile models.*

Lateral Load/Deflection (Head)

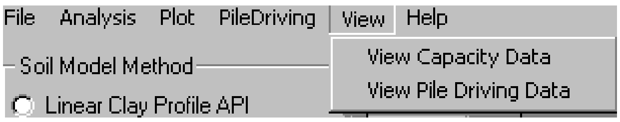
Axial Load/Deflection (Head)

Moment/Rotation (Head)

Pile Driving Menu -



View



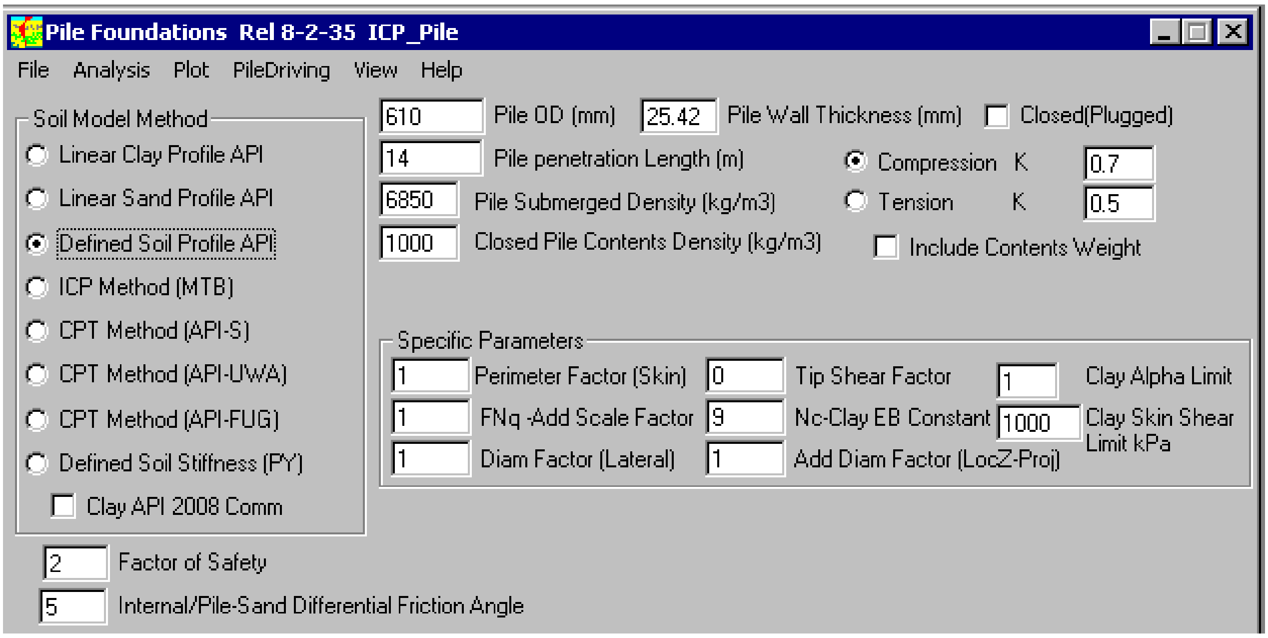
View Capacity Data Displays depth profile data, capacities & PY data etc.

View Pile Driving Data Display summary of pile driving analysis

-o-

3.2 Pile Method & Properties

This is the main window form of the program.



The **Soil Model Method** box provide the soil properties definition options. These options are

Linear Clay Profile API In this method the soil is assumed to be clay with a linear shear strength profile. When this option is active the Linear Clay Profile input box becomes visible. See Section 4.1.1 for method description.

Linear Sand Profile API In this method the soil is assumed to be sand. When this option is active the Piles in Sand properties input box becomes visible. See Section 4.1.1 for method description.

Defined Soil Profile API Properties are defined using a depth profile. Required data and format is given in Section 5.1.

ICP Method (MTB) Properties are defined using a depth profile. Required data and format is given in Section 5.2. See Section 4.1.2 for method description

CPT Method (API-S) Properties are defined using a depth profile. Required data and format is given in Section 5.2. See Section 4.1.3 for method description

CPT Method (API-UWA) Properties are defined using a depth profile. Required data and format is given in Section 5.2. See Section 4.13 for method description

CPT Method (API-FUG) Properties are defined using a depth profile. Required data and format is given in Section 5.2. See Section 4.1.3 for method description

Defined Soil Stiffness (PY) In this method the soil properties are specified by stiffness curves i.e. P-Y, T-Z and Q-Z curves. In this method the evaluated pile capacity is based on the ultimate capacities of the respective curves. The number of elements for the pile segment is defined by the user (prompted). The defined P-Y data will be interpolated to align with the element divisions. The P-Y curve data required and the data format is given in Section 5.2.

Clay API 2008 Comm Properties are defined using a depth profile. Required data and format is given in Section 5.1. See Section 4.1.4 for method description

The **Pile OD** and **Wall Thickness** define the pile. When the OD is input the wall thickness will be entered automatically using a D/t ratio of 30. The OD and t are used to evaluate the stiffness properties of the pile. The OD and t are also used to evaluate the pile self-weight used in capacity evaluation. It should be noted that the OD and t could be changed in FS2000 so that the Non-Linear pile stiffness analysis can be undertaken using different properties. Cases where this would be done is where the wall thickness of the pile changes with depth or in cases where the pile OD is not the pile/soil interface i.e. a drilled and grouted pile.

The **Pile Penetration Length** defines the depth of the pile. Pile capacities and other depth dependent properties will use this depth as a list or plot limit. When the pile model is created this depth will define the pile depth.

The **Factor of Safety** is used to evaluate the maximum load capacity of the pile. This value will be output at the top of the text output. The capacities used in the plots do not include this value.

The **Closed Pile Contents Density** is used to define the internal contents of the pile if closed (plugged). It is only used when evaluating the maximum pile capacity it is not used in the pile stiffness analysis. If the pile is plugged the value will be used. If the pile is not closed (plugged) the contents weight will be based on the external soil densities.

The **Internal/Pile-Sand Differential Friction** is used to define the difference between the internal angle of friction of the sand (ϕ) and the angle of friction between the pile and the sand (δ). $\delta = \phi$ - Differential Friction

The **Compression K** and **Tension K Factors** are use to evaluate the friction capacity for pile in sand (See Sect 4.1.1). The option buttons are used to determine which skin friction values are to be listed or plotted. The axial capacities show both tension and compression capacities regardless of the option buttons. If the Tension button is active the Q-Z spring will not be included in the structural model of the pile. The **Comp'n Only Tip** is a more realistic option for this purpose but it only works with the 3-D pile analysis.

If the **Clossed (Plugged)** check box is active the end bearing capacity will be based on the full area of the pile i.e. the pile is closed ended. If not, the end bearing will be based on the annular area and the internal plug (reverts to full area when the plug resistance > bore area end bearing resistance). Note that for the ICP method the pile is either open or closed ended and for the API CPT methods it is always open.

The **Include Contents Weight** simply includes/excludes the pile contents weight from the listed pile capacity - not normally included and usually only for tensile loads evaluation.

The following relate to the ICP method.

Specific Parameters

The following parameters can be used to enable the program to accommodate not standard piles e.g. suction cans. For normal driven piles these values should be left in their default condition.

The can be used to change the effective external area with respect to lateral resistance.

The **Add Diam Factor (Loc Z Proij)** can be used to change the effective external surface area with respect to lateral resistance in the direction parallel to the local Y axis of the element. The effective factor applied in this direction is the product of two lateral diameter factors as the **Diameter Factor (Lateral)** is applied to both axes. It can be used to model non-symmetrical piles.

The **Perimeter Factor (Skin)** can be used to increase the effective skin friction area with respect to axial shaft resistance. This can be used to account for effects of stiffeners

The **FNq Add Scal Factor** can be used to factor the Nq value for cohesionless soils so the value other than the API Nq can be applied.

The **Tip Shear Factor** can be used to provide an additional soil spring to represent lateral tip shear. The spring stiffness is based on a cubic parabola using the Q-Z quake values. For clays the shear capacity is based on the tip area x soil shear strength x Tip Shear Factor. For sands the capacity is based on the tip area x Tip End Brg x Tan(Soil Angle). The Tip End Brg is taken as the ultimate end bearing capacity / FoS (probably non-conservative). Ensure that the Tip Shear Factor is adjusted to suit the design assumptions and active loading especially with sand soils.

The **Nc-Clay EB Constant** defines the end bearing constant for cohesive soils. 9 is the normal API value for conventional piles.

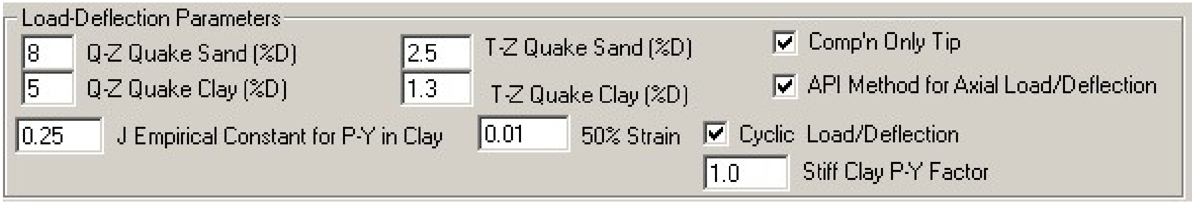
The **Clay Alpha Limit** limits the API clay Apha factor to a defined value.

The **Clay Skin Shear Limit** limits the API clay skin friction to a defined limit.

-o-

3.3 Soil Stiffness Properties

The Load-Deflection Parameters in the lower frame are used to define data relating to the evaluation of the soil stiffness curves e.g. P-Y curves.



.

If the **API** check box is active the Q-Z and T-Z curves will be evaluated in accordance with API other wise they will be evaluated in accordance with the alternative methods defined in Sects 4.2 and 4.3.   The quake value input boxes are used when the alternative methods are used.

If the **Cyclic** check box is checked the P-Y curves will be generated for cyclic loading.

The **J Constant** and the **50% Strain** are used in clay P-Y curve generation See Sect 4.5. These are the global default values. They may be defined as a function of depth using the extended profile data file (see Sect 5.1).

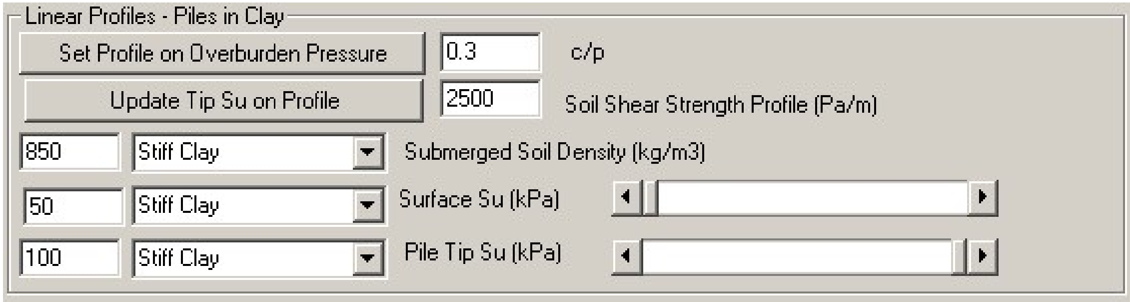
If the **Comp'n Only Tip** box is checked then the pile tip spring (Q-Z) will act as a compression only spring (3-D analysis only).

The **Stiff ClayP-Y Factor** is used to factor the PY curve for stiff clays under cyclic loading conditions. A stiff clay is a clay with a shear strength greater than 96 kPa. If a value < 1.0 is defined then the factor will Method B which used this factor will be use to construct the reduced strength P-Y curve. If value of =>1 is defined then Method A (modified Matlock) will be used to construct the P-Y curves. These methods are described in Section 4.5.

-o-

3.4 Linear Clay Properties

When the Linear Clay Profile option is selected the following input box becomes visible. This input form is used to define the properties of a cohesive soil.



The surface (pile head) and pile tip shear strengths are entered to define the linear soil profile.

When a value is entered in the box, the clay description box and the position of the slider bar will show its broad classification. For sensitivity studies the slider bars and clay type category boxes may be used to enter data for the shear strengths. Note that data entered directly into the box takes precedence and can be outside the range of the slider bars.

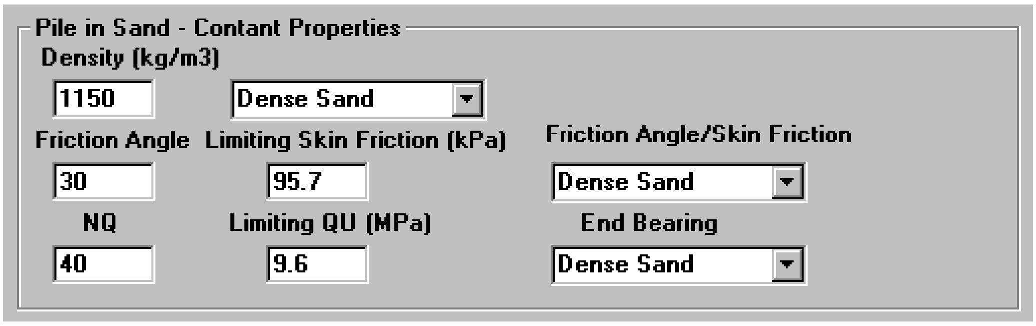
The submerged density is used to evaluate the overburden pressure.

The **Set Profile on Overburden Pressure** button can be used to define the soil shear strengths based solely on the soil submerged density. It does this by assuming that the ratio shear strength to overburden pressure is constant. The constant is shown in the **c/p** box.

-o-

3.5 Linear Sand Profiles

When the Linear Sand Profile option is selected the following input box becomes visible. This in form is used to define the properties of a non-cohesive soil.



The sand categories shown are those from API as given in Section 4.4.4. The friction angle defined in this form is the pile/sand friction angle.

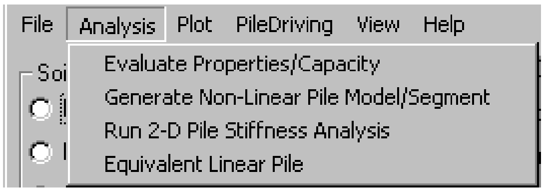
Data outside these categories may also be entered.

Note that although not used the sand friction listed in the output will include the addition of any specified **Internal/Pile-Sand Differential Friction.**

-o-

3.6 Analysis Menu

This menu is used to perform all analysis tasks related to the pile data.



Evaluate Properties/Capacities — Evaluates pile properties and load capacities. This includes the stiffness pile stiffness properties i.e.the P-Y data etc. and the element soil spring file (UNP).

Generate Non-Linear Pile Model — Evaluations as previous command but also creates an FS2000 model geometry definition file (UM file) and the element/soil spring association file (UPN).

Run Pile Stiffness Analysis — Creates and runs 2-D load cases (pile head load conditions). Pile model must have first been saved in FS2000.

Equivalent Linear Pile Element — Specifies an equivalent linear beam based on specific loading conditions

-o-

3.6.1 Evaluate Properties/Capacities

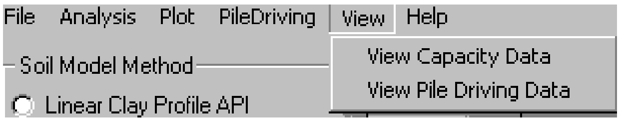
This menu option generates the pile capacities, the pile stiffness characteristics (P-Y data etc) and element soil spring properties.

The soil spring property data used in the solution is contained in the UNP file.

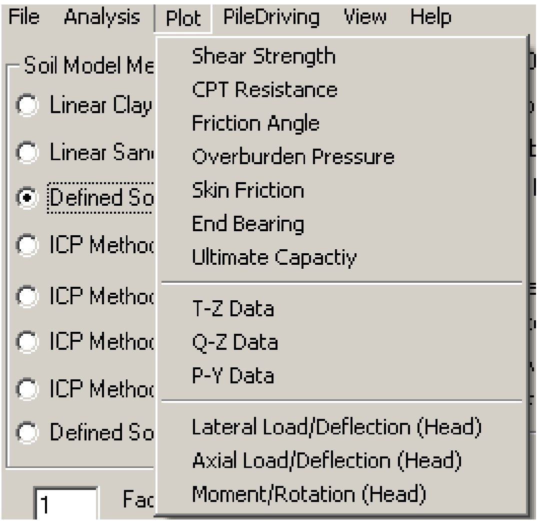
This spring data is used during a non-linear solution and it specifies the axial and lateral soil stiffness spring at the element mid point. An additional tip point is present at the bottom of the pile. This is used to represent pile end bearing resistance and the optional lateral tip shear resistance.

If the soil definition method uses User Defined soil data then the program will prompt for the file name of the data. It is recommended that the default naming convention (see Section 6.0) be adopted.

When this operation is completed the capacity & soil stiffness data file will be created. This file can be viewed/printed using the **View** menu. The pile results data shown is in a .DPR file.



The results of this process can also be plotted using the following plot options.



Use FS-Edit to view the UNP data file.

-o-

3.6.2 Generate Non-Linear Pile Model

This menu option is used to generate the non-linear isolated pile or a pile segment for an extended pile model.

This generation process creates the following files:

UM1 Command file containing the pile model definitions file. Requires later interpretation in FS2000's GUI.

UPM This UPM file associates the elements with the springs - used by solvers.

This option also re-evaluates the Properties/Capacities by activation **Evaluate Properties/Capacities** command to ensure springs relate the most recent generated model data.

The is an option to use Type 7 couples to represent the non-linear spoil springs. If the option is adopted the spring data is included in the UM file.

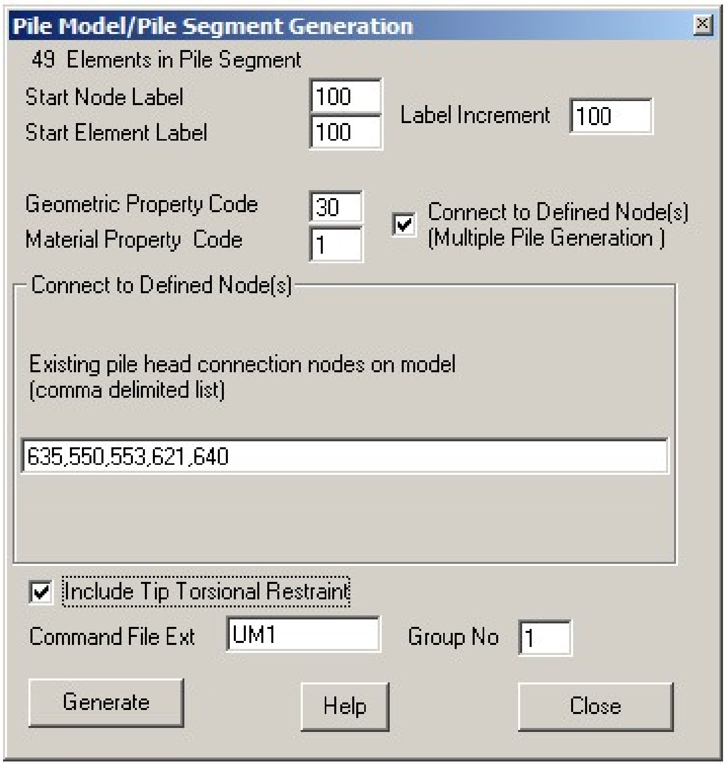
To include a pile segment in an FS2000 model, the .UM1 file must be interpreted within Model Generation TASK of FS2000.

To add additional piles to a model, simply create a additional .UM files (UM2 , UM3 etc) using different node and element start labels.

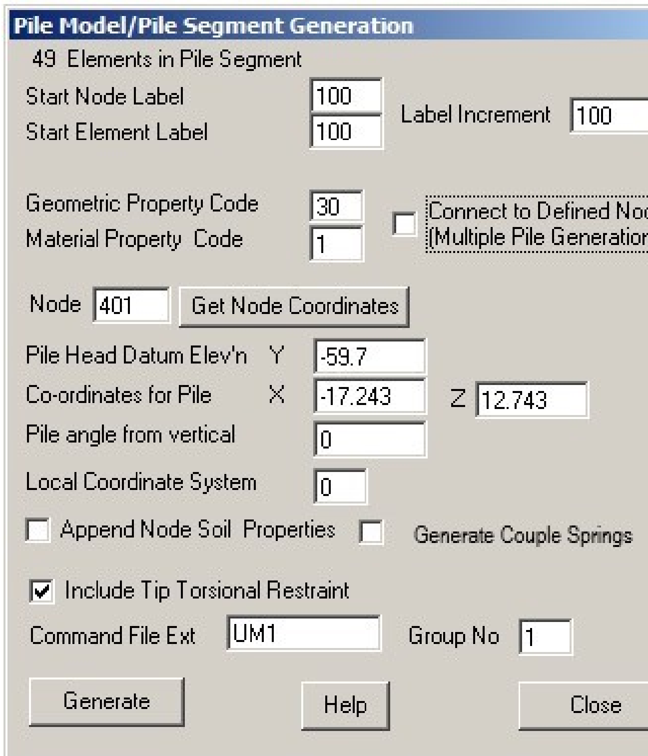
The following form is used to created the .UM files. There are two basic modes of operation, Single and Connection. The **Connect to Defined Node(s)** option switches between the two model of operation.

The Connection mode is the recommended mode of operation because in will create a single UM file that will connect one or more piles to existing model nodes. It also retains the setting so that ant subsequent regeneration of pile segment is a one click operation.

Connection Mode



Single Mode



The top of the form states the number of elements that will be created in forming the pile segment model.

The **Start Node Label** and the **Start Element Label** are used to define what labels will be assigned to the pile. The start node label will be at the top of the pile. When generating additional piles or pile segments for an existing model ensure that the existing pile nodes and elements are not overwritten. If the pile model is to be used to generate properties of an equivalent linear pile the Start Nodel (Pile Head) label must be 1.

The **Label Increment** can be used to set a fixed increment between generated pile segments. Always make this larger then the number of elements in the pile segment and allow a margin if the size is likely to increases.

The **Geometric** and **Material Property Codes** are used to define the properties of the pile in the FS2000 model. The material properties will default to a Grade 50 steel.

The **Include Tip Torsional Restraint** will added the necessary torsional restraint for vertical pile (not requireded other restraint is provided).

The **Generate Couple Springs** option will generated Type 7 Couples (with RC properties) to represent the non-linear soil springs - see Section 5.4.1, This can only be used with **Single Mode Operation**/Generation.

The **Command File Ext** is the command file created with the pile is generated. Not the only on file is created when the Connect Mode is used.

The **Group No** is the element group assigned to each generated pile segment.

Connection Mode Operation

The node list box is used to define the existing model nodes to which the pile segments will be connected. Up to 16 piles can be connected in one generation operation. The list is comma delimited.

The above node list: **635,550,553,621,640** and settings would generate 5 pile segments starting with element and node labels 100, 200, 300, 400 ,500

Only one UM file is ceated in this mode of operation.

In this mode of operation all generation setting are stored which makes the regeneration of different pile sizes were easy.

Single Mode Operation

The **Local Coordinate System** can be use to map the pile to existing model coordinate systems. It does by equating the pile length direction to the local x directions. The model coordinate system would normally be set with the local x axis vertical and starting at the point where the pile head is located. If a non-zero value is entered the coordinates will mapped.

The **Pile Head Datum Elev'n (Y)** and **X Co-ordinate** and **Z Co-ordinate** are used to define the location of the pile in terms of the model co-ordinate system. Note that the local coordinate system method is more convenient if pile generation is to be repeated.

The **Get Node Coordinates** button is used to get the coordinates of a an existing node. This is useful when adding piles to an existing model.

It is not essential to use these values since the pile may be transformed in FS2000.

The **Append Soil Properties** should only be checked when additional piles are to be generated i.e. more than one UM file is to be created. This option controls the file that assigns the appropriate soil springs to the respective elements. This association is stored in a .UPN file. For the first pile segment this should be unchecked (.UPN will be created) for subsequent pile segments this should be checked (data will be appended to the .UPN).

Piles with Different Soil Properties

It is not currently possible to assign different soil properties to different piles in the same model using FS-Pile directly but it can be done by merging and editing UNP and UNP files. An approach using individual pile models would be:

- For each different soil profile (from primary model) create an isolated pile model. When generating the pile element in each model use a appropriate node and element label increment as would be done in a single model.
- In the second and subsequent pile models edit the UNP soil reference number i.e renumber so that they will sequential across the pile model.
- In the second and subsequent pile models edit the UPN file so that the elements are associated

with the renumbered UNP data..

- Merge the UNP and UPN files together in the primary model ensuring that the format is correct, see Sect 5.4.
- Interpret each the UM files into the primary model.

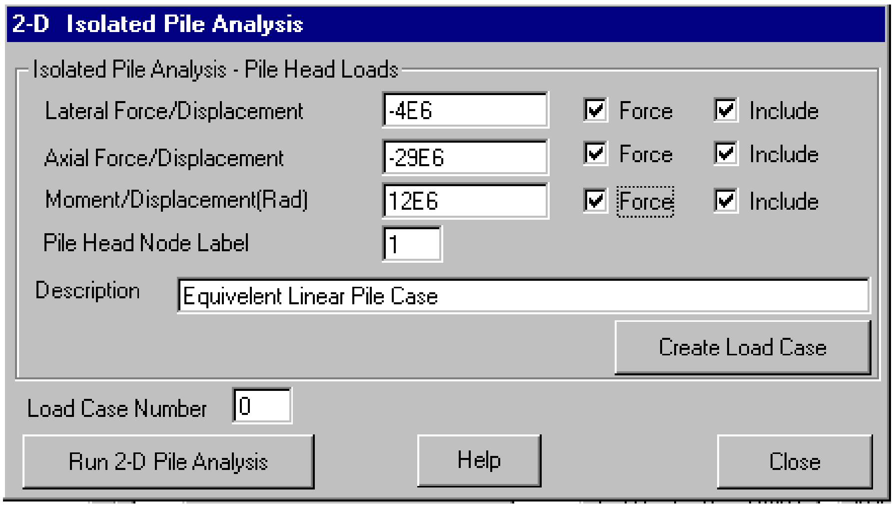
The above may also be undertaken within one model. This approach would require the UNP file and UPN file to be ren-named after each pile segment generation. The UNP and UPN files could then be edited and merged. This approach is a better option because all definition data can be archived in one model (note that extension names must be preceded with a U e.g. UNP1 to be archived). Note that when using this approach there would be a different USP file for each different soil profile.

-o-

3.6.3 Run Pile Stiffness Analysis

This command option is used to run both 2-D isolated and 2-D extended non-linear pile models. The form that appears when this option is selected enables pile head load or displacements to be applied to the pile model.

Note that the pile UM file must have first been interpreted and the pile model save in FS2000 prior to running a pile analysis.



Pile Forces or displacements for isolated pile are entered in the input boxes. The **Force** check box is used to indicate that the value is a force. The **Include** check box must be checked to include the value in the load case to be created.

The **Create Load** case button will generate an isolated pile load case. It will use the Load Case Number and Description currently entered.

The **Load Case** is used to define the load case number to be created or run. Note that loads cases created on FS2000 can also be run from here.

The **Analysis** button will run the load case entered in the **Load Case** box. Use this to run both isolated and extended pile model load cases. When the analysis is run the load case defined will be divided into 10 equal increments. The load will be progressively applied to the pile head in these load increments. This enables the load/displacements characteristics to be obtained.

As the solution progresses the number of iteration to convergence for each load set will be shown. If convergence is not achieved in 10 iterations in any load step than the solution for that load step is likely to be invalid and all subsequent load step solution will also be invalid. This indicates that the limit load level has been exceeded. Lower the load level and repeat.

Running Pile Analysis in FS2000's GUI

The following **Solution** command can be used to run pile models.

3-D Non-Linear
2-D Pile Analysis
DyNoFlex

Running Pile Models in Batch

A pile model is just like any other non-linear model and can be run in batch mode,

-o-

3.6.4 Equivalent Linear Pile

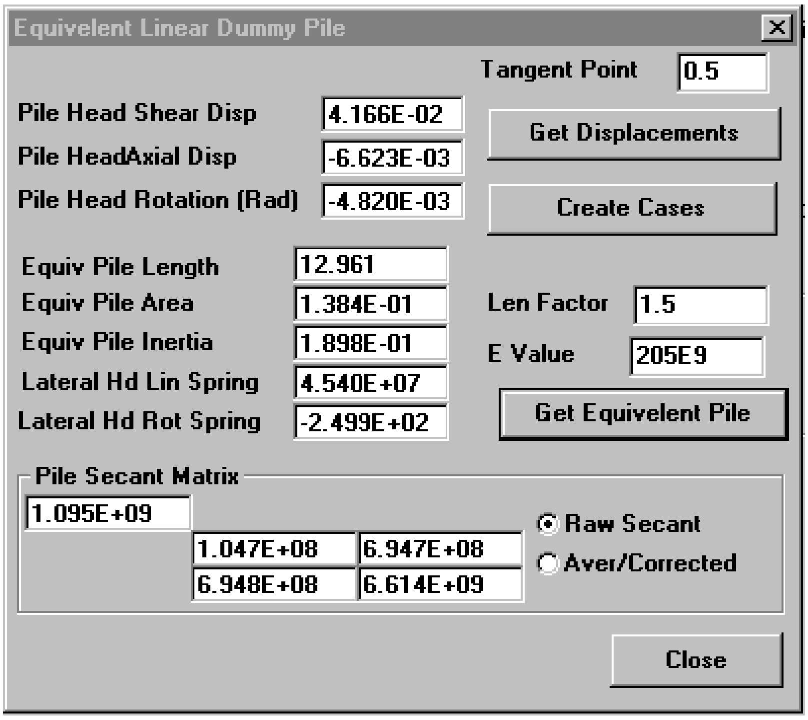
This command option is used to evaluate the properties of an equivalent linear pile at pre-defined load levels.

To use this function the pile head node label MUST be 1.

To use this option is mandatory to create and run Load Case 0 with the Head Lateral Force, Head Axial Force and Head Moment at the load levels at which the equivalent properties are required. See Section 3.6.3

Note: Due to the sign convention used the Lateral Force should be the opposite sign to the Moment.

The form that appears when this option selected is given below.



The **Get Displacements** button obtains the displacements at a point of the of the load characteristics of Load Case 0. The default value for this point is 0.5 i.e. mid point. The values obtained are shown in the appropriate boxes.

The **Create Cases** button is used to create the necessary load cases to evaluate the secant stiffness matrix at the point defined by the **Tangent Point** box. The default for this value is 0.5 i.e. mid point. Once the load cases have been created they require to be run. A batch file (BRM) is automatically created to run the 4 cases. Use the Batch Run Module to run these.

When the load cases have been run the **Get Equivalent Pile** button is used obtain the equivalent pile properties. The Len Factor should be adjusted so that positive values are obtained. This value is usually in the range 1.5-2.5. This value can be tuned to eliminate the pile rotational spring.

Note that the values obtained for the Get Displacements must be visible for the pile data to be evaluated.

The secant matrix coefficients can be viewed below. Both the raw coefficients and the average/corrected coefficients can be viewed. See Section 4.6 for background theory.

-o-

3.7 Creating and Running Extended Pile Models

Pile elements can be added to any model. The only consideration is that when pile elements are added to a model the do not overwrite existing nodes or elements in the model. This means that to accommodate the addition of pile elements a block of element and node label ranges have to be reserved for the piles. It not important where the range is and it not necessary that all pile elements are in the same block. It is probably more convenient to put the at the front end of a model. If say 4 x 50 element piles were to be used, then the model first non pile nodes and elements would start at 250.

The following procedure using the **Single Mode** approach is typical for creating an extended pile model (with two piles) is;

- Define the Soil Property data e.g. USP file.
- Save the pile data (File menu/Save command)
- Generate the pile properties/capacity - This generates the UNP soil spring file and the pile capacity data.
- Use the Generate Non-linear Pile Model/Segment command and generate the pile UM file and associated stiffness data.
- Interpret the .UM command files in FS2000's GUI and connect the piles to the structure.
- Modify the model as appropriate and save the model in FS2000.

The model can bow be run one of the non-linear solvers and the results processed like any other model.

Solution of Models with Piles

The standard linear solution module of FS2000 cannot be used with pile models since they do not recognise the non-linear pile springs.

The only solvers that can be used are:

- The **Run Pile Stiffness Analysis** command in FS-Pile used for 2-D pile models
- The **3-D Non-Linear**, the **2-D Pile Analysis** or **DyNoFlex**

Only two pile related files are used in the solution. The UPN file which associates Type 6 element with soil spring data and the UNP file which contains the soil spring data.

Additional restraints can be applied to pile models so that they can analysed with the linear solver. This is sometimes useful for debug purposes if a model is not running for correctly for reasons other than pile supports i.e. fully restrain the pile nodes at a point just below the surface It the additional restraints are removed the model can then be run as a pile model.

Sensitivity Studies

If the length of the pile is changed in FS-Pile it is necessary to re-generate the pile model in FS2000. This is a requirement since the number of nodes and elements in the pile segment will change as the length changes.

In such cases the existing pile nodes and elements should be deleted prior to re-interpreting the UM file.

If the soil properties are the only changes then it is not necessary to re-interpret the UM files into the model because the **Evaluate Properties/Capacities** will update the UNP file**.**
If the wall of the pile is changed only the property code in FS2000 need be modified for the structure solution. OD changes require pile spring re-evaluation.

-o-

4.0 Background Pile Theory & Methods

4.1 Pile Load Capacities

This presents the methods and equations used to evaluate pile load capacities and pile stiffness characteristics.

The program will produce design capacities using a user defined Factor of Safety (FOS). The design capacity is based on the assumption that the pile weight is an active load rather than a reduction in pile resistance. The following illustrates how the design load is evalauted.

Compressive Design Capacity = Ultimate Capacity / FOS - Pile Weight

Tensile Design Capacity = Ultimate Capacity / FOS + Pile Weight

If pile contents weight is included this would be added to the pile weight. This would normally be only specified for tensile load capacity evaluation.

Note that pile and contents weights do not effect the pile stiffness characteristics i.e. the structural model should include all load effects.

-o-

4.1.1 API Methods

Ultimate Static Capacity Based on API RP2A

The ultimate capacity of an open-ended pipe is given by:

Qult = fs.As + fi.Ai + qp.Aw

fs.As = outside shaft friction

fi.Ai = inside shaft friction

qp.Aw = end bearing on pipe wall area

If the internal friction exceeds the end bearing capacity of the end plug the pile behaves as a closed ended pipe (plugged pile) in which case

Qult = fs.As + qp.Ap

qp.Ap = end bearing on gross pipe area

Ultimate Capacity in Cohesionless Soils (Sands)

Shaft Friction = f = po.K.Tan δ (β = K.Tan δ where K=0.8 in 2008 Supplement)

End Bearing = qp = po.Nq

K = Coefficient of lateral earth pressure (API suggest 0.8 for tension and compression)

po= Effective overburden pressure

δ = Angle of Friction between soil and pipe =ϕ - Differential friction angle

Nq = Bearing Capacity Factor

Density	Soil Type	Soil-Pile Friction Angle Degrees	Limiting Friction kPa	Nq	Limiting qu MPa
Very Loose	Sand	15	47.8	8	1.9
Loose	Sand-Silt				
Medium	Silt				
Loose	Sand	20	67	12	2.9
Medium	Sand-Silt				
Dense	Silt				
Medium	Sand	25	81.3	20	4.8
Dense	Silt				
Dense	Sand	30	95.7	40	9.6
Very Dense	Silt				
Dense	Gravel	35	114.8	50	12
Very Dense	Sand				

Ultimate Capacity in Cohesive Soils (Clays)

Unit Shaft Friction = ac

Unit End Bearing = 9c

c = Undrained soil shear strength

a = A dimensionless factor   <= 1.0

a = 0.5 (c/po)**-.5   for c/po <= 1.0

a = 0.5 (c/po)**-.25   for c/po > 1.0

po = overburden pressure

-o-

4.1.2 ICP (MTD) Methods

These methods are based on 'New Design Methods for Offshore Piles', MTD (96/103). This document summarises the method developed at Imperial College, London.

SANDS Shaft Capacity of closed-ended piles

$Q_s = \pi D \int \tau_f \, dz$	Shaft capacity
$\tau_f = \sigma'_{rf} \tan \delta_{cv}$ $\sigma'_{rf} = (\sigma'_{rc} + \Delta\sigma'_{rd})$	Local shear strenght
$\sigma'_{rc} = 0.029\, q_c\, (\sigma'_{v0}/P_a)^{0.13}\, (h/R)^{-0.38}$	Local Radial Shear Stress A lower limit of h/R = 8 applies Pa = 100 kPa
$\Delta\sigma'_{rd} = 2G\, \Delta r / R$ $G = q_c\, [A + B\eta - C\eta^2\,]^{-1}$ where $\eta = q_c\, (P_a\, \sigma'_{v0})^{-0.5}$ $A = 0.0203$ $B = 0.00125$ $C = 1.216e\text{-}6$	Dilatant increase in local radial effective stress during pile loading Δr = 2E-5
δ_{cv} Measure directly in tests. If not feasible estimate from Figure 5	Interface angle of friction at failure
$\tau_f = (0.8\sigma'_{rc} + \Delta\sigma'_{rd})\, \tan \delta_{cv}$	Tension loading

SANDS Shaft Capacity of open-ended piles

$\sigma'_{rc} = 0.029\, q_c\, (\sigma'_{v0}/P_a)^{0.13}\, (h/R^*)^{-0.38}$ $R^* = (R^2_{outer} - R^2_{inner})^{0.5}$	A lower limit of h/R* = 8 applies
In tension $\tau_f = 0.9(0.8\sigma'_{rc} + \Delta\sigma'_r)\, \tan\, \delta_{cv}$	

SANDS Base Capacity of Closed-ended Piles

$Q_b = q_b \pi D^2/4$ $q_b = q_c [1 - 0.5 \log (D/D_{CPT})]$	qb = 0.13qc for D > 2m

SANDS Base Capacity of Open- ended PIles

$D_{inner} < 0.02 (D_r - 30)$

$D_{inner}/D_{CPT} < 0.083 q_c/P_a$

A rigd basal plug can only develop if this criteria is satisfied. Not that Dr is specified in %.

$Q_b = q_b \pi R^2_{outer}$ $q_b = q_c [0.5 - 0.25 \log (D/D_{CP}$	Full pugged piles
$Q_b = q_{ba} \pi (R^2_{outer} - R^2_{inner})$ $q_{ba} = q_c$	Unplugged and large open ended piles

CLAYS Shaft Capacity of closed-ended piles

$Q_s = \pi D \int \tau_f \, dz$	Shaft capacity
$\tau_f = \sigma'_{rf} \tan \delta_f = (K_f/K_c)\, \sigma'_{rc} \tan \delta_f$	Local shear strenght Kf/Kc = 0.8
$\sigma'_{rc} = K_c\, \sigma'_{v0}$ $K_c = [2.2 + 0.016\, YSR - 0.870\, \Delta I_{vy}] YSR^{0.42}\, (h/R)^{-0.20}$ and $\Delta I_{vy} = \log_{10} S_t$	Local Radial Effective Stress
δ_f between δ_{peak} and $\delta_{ultimate}$	Interface angle of friction

CLAYS Shaft Capacity of open-ended piles

$K_c = f(h/R^*)$ where $R^* = (R^2_{outer} - R^2_{inner})^{0.5}$	A lower limit of h/R* = 8 applies Kc as above

CLAYS Base Capacity of Closed-ended Piles

$q_b = 0.8\ q_c$	Undrained loading	
$q_b = 1.3\ q_c$	Drained loading	

CLAYS Base Capacity of Open- ended PIles

$[D_{inner}/D_{CPT} + 0.45\ q_c/P_a] < 36$ — Plugging takes place if criteria is satisfied

$Q_b = q_b\ \pi\ D^2/4$		Full pugged piles
$q_b = 0.4\ q_c$	Undrained loading	
$q_b = 0.65\ q_c$	Drained loading	
$Q_b = q_{ba}\ \pi\ (R^2_{outer} - R^2_{inner})$		Unplugged piles
$q_{ba} = q_c$	Undrained loading	
$q_{ba} = 1.6 q_c$	Drained loading	

-o-

4.1.3 CPT (API) Methods

The API CPT based methods implemented in FS-Pile are the Simplified ICP 05 (API-S), the Offshore UWA-05 (API-UWA) and Fugro 05 (API-FUG) methods . These methods are only applicable to cohesionless soils. For stratified soils the API method for clay layers will be implemented using the API method or the API Commentry method.

$$Q_d = Q_f + Q_p = P_o \int f_{c,z} dz + A_p q_p$$

$$Q_t = P_o \int f_{t,z} dz$$

Skin Friction

The unit skin friction (f) formulae for open ended steel pipe piles for the first three recommended CPT-based methods (Simplified ICP-05 **,** Offshore UWA-05 and Fugro-05) can all be considered as special cases of the general formula:

$$f_z = u \cdot q_{c,z} \left(\frac{\sigma'_{vo}}{p_a} \right)^a A_r^{\,b} \left[\max\left(\frac{L-z}{D}, v \right) \right]^{-c} \left[\tan\delta_{cv} \right]^d \times \left[\min\left(\frac{L-z}{D} \frac{1}{v}, 1 \right) \right]^e$$

Where:

Ap	Pile gross end area = πDo**2/4
Ar	Pile displacement ratio 1-(Di/Do)**2
Do	Pile outer diameter
Di	Pile inner diameter = Do - 2t
t	is the wall thickness of the pile
L	Pile embedded length (below original seabed)
pa	Atmospheric pressure = 100 kPa
qc,z	CPT cone tip resistance qc at depth z
σ'v0	is the overburden pressure at depth z

Values for parameters a, b, c, d, e, u and v for compression and tension):

Method	**Parameter**						
	a	b	c	d	e	u	v
Simplified ICP-05							
Compression	0.1	0.2	0.4	1	0	0.023	$4\sqrt{A_r}$
Tension	0.1	0.2	0.4	1	0	0.016	$4\sqrt{A_r}$
Offshore UWA-05							
Compression	0	0.3	0.5	1	0	0.030	2
Tension	0	0.3	0.5	1	0	0.022	2
Fugro-05							

Compression	0.05	0.45	0.90	0	1	0.043	$2\sqrt{A_r}$
Tension	0.15	0.42	0.85	0	0	0.025	$2\sqrt{A_r}$

End Bearing

Simplified ICP-05

This method uses the same approach as the ICP(MTB).

Offshore UWA-05

$$q_p = q_{c,av,1.5D}(0.15 + 0.45A_r)$$

Ar = Pile displacement ratio 1-(Di/Do)**2

Fugro-05

$$q_p = 8.5p_a\left(\frac{q_{c,av,1.5D}}{p_a}\right)^{0.5} A_r^{0.25}$$

The criterion for plugging is that the thickness of the sand plug within the pile is greater than 8D or

$$Q_p \le Q_{f,i,clay}e^{L_s/D}$$

This criterion in not checked by FS-Pile.

-o-

4.1.1 API Clay Comm Method

This method is the alterantive method for clays presented in the commentary of API RP2A (2008 Supplement).

Skin Friction

The skin friction f is given by:

$$f = c_u \qquad \text{for } c_u < 0.5 \text{ kips/ft}^2 \text{ (24 kPa)}$$

$$f = c_u/2 \qquad \text{for } c_u > 1.5 \text{ kips/ft}^2 \text{ (72 kPa)}$$

Interpolation is used for intermediate values of Cu

End Bearing

This method uses the same approach as the API Clay main text method.

-o-

4.2 T-Z Curves in Sand and Clay

API Method in Sand and Clay

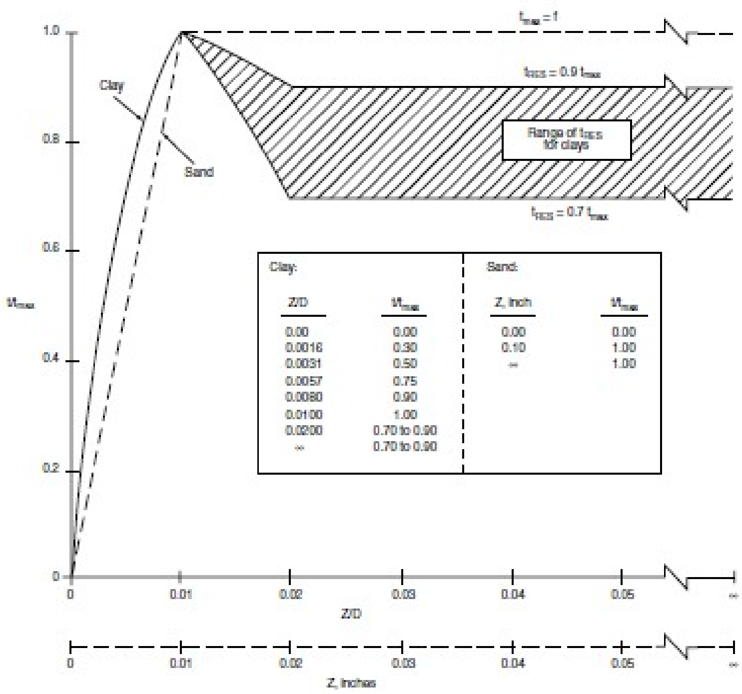
In the API method the quake values are implicitly defined with the recommended T-Z curve.

For sand a quake value of 2.54 mm is employed and the following curve is constructed.

Z/Zu	T/Tu
0.250	0.250
0.500	0.500
0.700	0.700
0.850	0.850
1.000	1.000
10.00	1.000

For clay a Quake value of .01D is employed and the following curve is constructed

Z/D	T/Tu
0.0016	0.30
0.0031	0.50
0.0057	0.75
0.0080	0.90
0.01	1.00
0.02	0.80



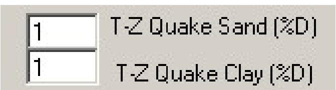
Alternative Method in Sand and Clay

The only additional properties required to generate the T-Z curve is the pile tip quake values.   These are defined in terms of a fraction of the pile diameter.

The method proposed by Vijayvergiya (1977) is the basis for the alternative method

T/Tu = 2(Z/Zu)**1/2 - Z/Zu

The quake values (Zu) are entered as a percentage of the pile diameter.



When the non API option is active the following table is used to construct the T-Z curves for both sand and clay.

Z/Zu	T/Tu
0.125	0.582
0.250	0.75
0.500	0.914
0.759	0.982
0.875	0.99
1.00	1

-o-

4.3 Q-Z Curves in Sand and Clay

API Method in Sand & Clay

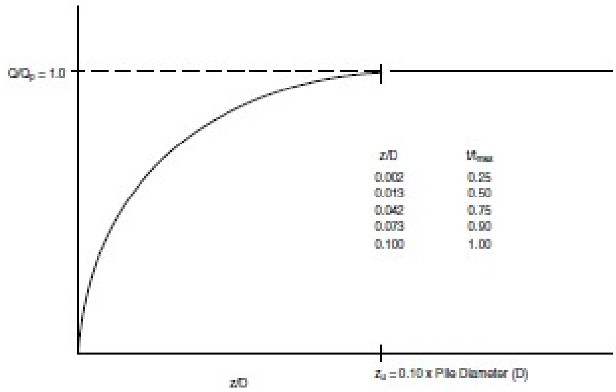
The only property required to generate the Q-Z curve is the pile tip quake value. This is defined in terms of a fraction of the pile diameter.

The API method suggests the cubic parabola (Vijayvergiya, 1977)

Q/Qu = (Z/Zu)**1/3

API uses a quake value of 10% for both sand and clay and recommends the following Q-Z curve. This is adopted if the API option is active.

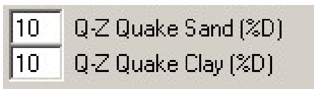
Z/D	Q/Qu
0.002	0.25
0.013	0.50
0.042	0.75
0.073	0.90
0.100	1.00



Alternative Method in Sand & Clay

If the API option is not activated the use can define different values for both sand and clay soils. In all other respects the method is identical to API.

The quake values (Zu) are entered as a percentage of the pile diameter.



When the non API option is active the following table is used to construct the Q-Z curve.

Z/Zu	Q/Qu
0.125	0.497
0.250	0.630
0.500	0.794
0.759	0.908
0.875	0.956
1.00	1.00

-o-

4.4 P-Y Curves in Sand

The soil properties required to generate P-Y curves are:

- effective soil density - generally a function of depth
- angle of friction - generally a function of depth

API Method in Sand

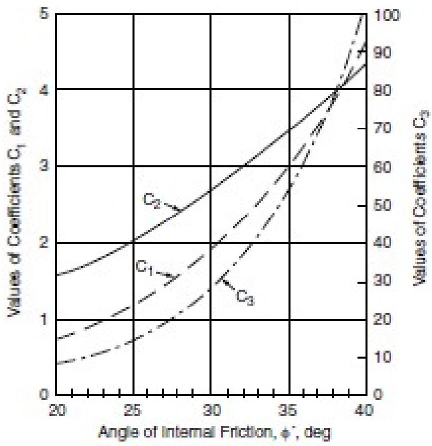
Lateral Ultimate Capacity

Evaluate pu (ultimate resistance) based on the depth relative to

At a given depth pu is the lesser of

pu = (C1X + C2D).γ.X

pu = C3.D.γ.X



Load - Deflection Curves

The load deflection curves are based on the following relationship

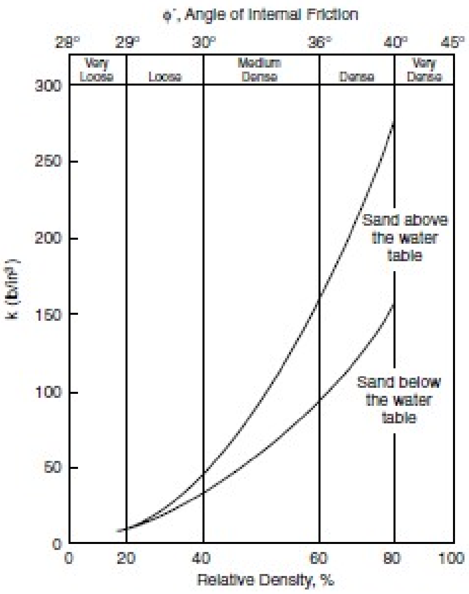
P = A.pu.tanh[(k.X/A.pu).y]

A = Factor to account for static or cyclic loading

A = 0.9 for cyclic loading

A = (3 + 0.8X/D) <= 0.9 for static loading

k = initial modulus of subgrade reaction (f'unction(ϕ internal friction angle)



The relationship is established at the following points

y = yu = 3D/80

y = 0.10yu

$y = 0.25y_u$

$y = 0.50y_u$

$y = 0.80y_u$

$y = 0.25D$

-o-

4.5 P-Y Curves in Clay

The soil properties required to generate P-Y curves are:

- effective soil density - generally a function of depth
- Undrained shear strength - generally a function of depth
- strain corresponding to 50 ultimate stress in clay
- J coefficient for evaluation of ultimate soil resistance in soft clay

API Method in Soft Clays

Soft clay are clays with c (undrained shear strength) less than 96 kPa.

Lateral Ultimate Capacity

Evaluate pu (ultimate resistance) based on the depth relative to Xr (depth to bottom of reduced resistance zone)

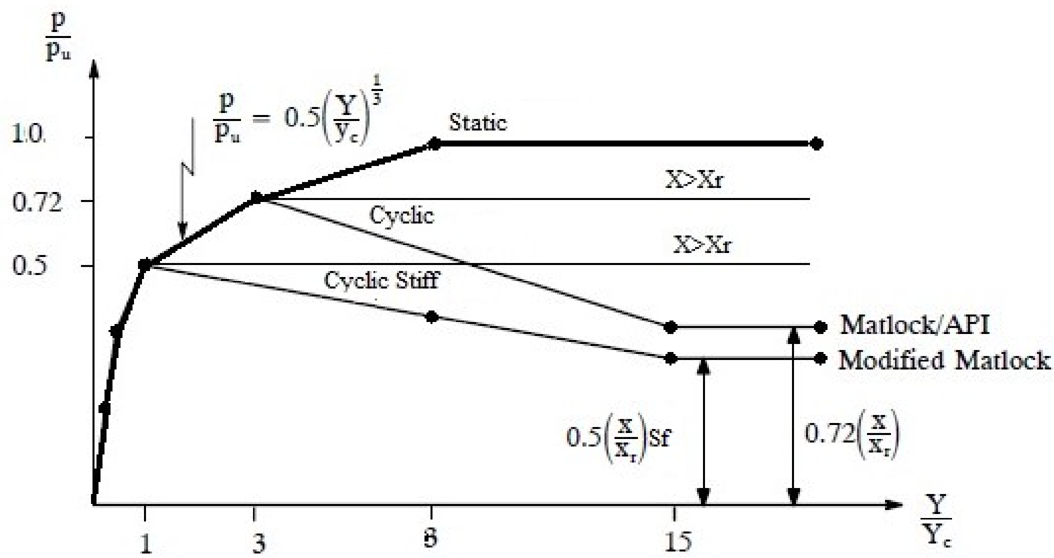
pu = 3c + γX + JcX/D for X < Xr

pu = 9c for Xr >= Xr

Xr = 6D/(γD/c + J)

J = 0.25 to 0.5 (use .25 in absence of specific data)

Load - Deflection Curves



The load deflection curves are based on the following relationships

yc = 2.5Dε

ε = strain corresponding to 50 ultimate stress

ε = 0.005 brittle/sensitive clay

ε = 0.02 remoulded/unconsolidated

ε = 0.01 Soft - generally most applicable

Method Adopted for Soft Clays (API)

Static in Soft Clay

p/pu	y/yc
0.23	0.1
0.33	0.3
0.500	1.0
0.720	3.0
1.000	8
1.00	18

(p/pu = 0.5(y/yc)**(1/3) for y/yc<= 8)

Cyclic in Soft Clay - Equilibrium Reached

X>Xr		X<Xr	
p/pu	y/yc	p/pu	y/yc
0.23	0.1	0.23	0.1
0.33	0.6	0.33	0.6
0.500	1.0	0.500	1.0
0.720	3.0	0.720	3.0
0.720	15	0.72X/Xr	15
0.720	18	0.72X/Xr	18

(p/pu = 0.5(y/yc)**(1/3) for y/yc<=3)

Methods Adopted for Stiff Clays

Stiff clays are clays with c (undrained shear strength) greater than 96 kPa.

Static in Stiff Clay - As for soft clays

Cyclic in Stiff Clays

Two options are available depending upon the define value of the **Stiff Clay P-Y Factor (Sf).**

Method A (Stiff Clay P-Y Factor => 1.0)

This is the Modified Matlock method (a Fugro adopted method with Sf=1.44).

X>Xr		X<Xr	
p/pu	y/yc	p/pu	y/yc
0.23	0.1	0.23	0.1
0.33	0.6	0.33	0.6
0.5	1.0	0.500	1.0
0.5	8.0	*	8.0
0.5	15	0.5X/Xr*Sf	15
0.5	18	0.5X/Xr*Sf	18

(p/pu = 0.5(y/yc)**(1/3) for y/yc<=3)

* Point 4 is replaced by a linear interpolation between point 3 and points 5.

Method B (Stiff Clay P-Y Factor < 1.0**)**

As for soft cyclic clays but with p/pu value being factored by the **Stiff Clay P-Y Factor** (Points 4,5 & 6 only).

This will give similar results to the modified Matlock method if a factor of 0.694 is employed but with a flat section between y/yc and 3y/yc.

This method will be adopted if the **Stiff Clay P-Y Factor** is defined as less than 1.0.

-o-

4.6 Equivalent Linear Piles

The program uses the following secant approach to evaluate an equivalent linear pile.

The Axial Load (Fy), Shear Load (Fx) and Moment (M) at the pile head corresponding to the full load condition at which the pile is to be equated are applied to the pile model.

The pile is than linearised at the mid load point. This is achieved by first applying a small lateral displacement X (rotation held constant at R/2) about the mid point so as to obtain Fx and M, and then applying a small rotation R (laterally fixed at X/2) about the mid point so as to obtain Fx and M. (4 Cases evaluated)

		Fy/Y			
			Fx/X	M/X	
{Ktr}	=				Raw Secant Matrix (Tangent - Unsymetrical)
			Fx/R	M/R	

This enables the following matrix to be formed.

		Fy/Y			
			Fx/X	1/2(Fx/R+M/X)	
{Kt}	=				Averaged Raw Secant Matrix
			1/2(Fx/R+M/X)	M/R	

A correction matrix is used to evaluate the characteristics at the full load.

{Fc}	=	{Fo}	-	{Kt}{Xo}	=	{Kc}	
{Kp}	=	{Kt}	+	{Kc}			Averaged/Corrected Matrix

The resulting matrix is then equated to matrix based on beam/spring properties.

		EA/L	0	0	
{Kpe}	=	0	12EI/L**3+kx	6EI/L**2	Averaged /Corrected Matrix
		0	6EI/L**2	4EI/L+kr	

If L = 1.5Kp33/Kp23

then kr = 0

and

kx = Kp22 - 12EI/L**3

-o-

5.0 File Formats - User Defined Data

This section describes the file formats which can be used to define user defined soil properties.

The first is the Defined API Soil Profile. This is used to define soil properties in terms of defined profiles ie as a function of depth for the normal API methods.

The second is the Defined CPT Soil Profile. This is used to define soil properties in terms of defined profiles ie as a function of depth for the CPT methods.

The third is the Defined Soil Stiffness Profile. This is used to define the soil properties in terms of it stiffness characteristics ie P-Y data etc.

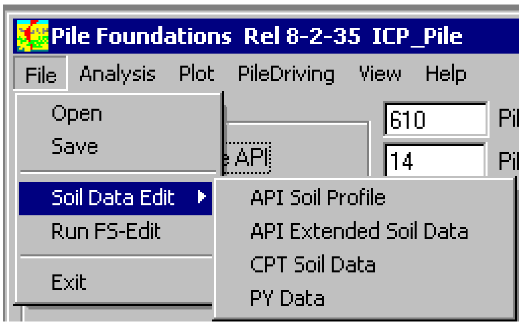
Data File Naming

The naming convention described in this section does not have to be strictly adhered to. If only one soil profile is to be used then there is no reason to change the defaults. If however more than one profile is to be used then the data file names can be extended to identify such

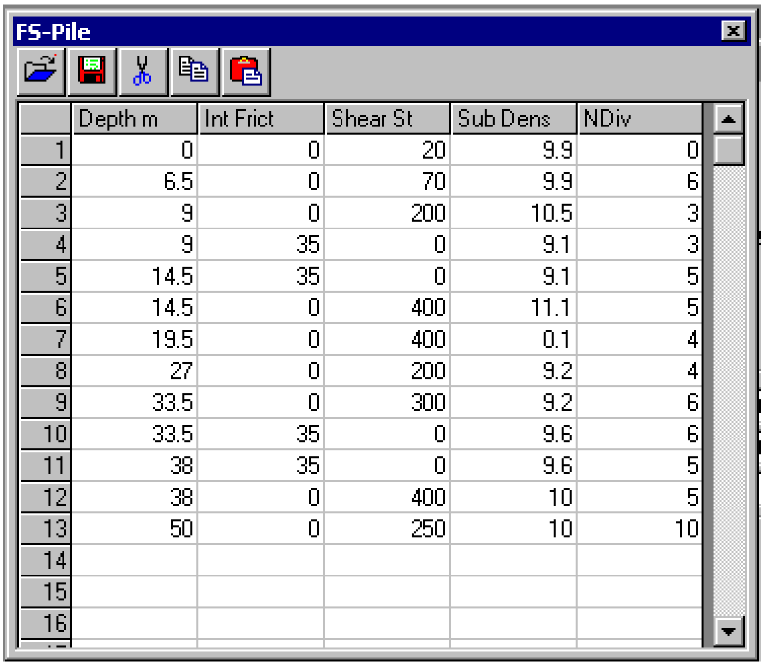
e.g <modelename).USPUpper and <modelename).USPLower could be used to identify two different soil profiles.

Grid Editors

The data files may be created in any text editor using the data formatting rules given in this section but for convenience a grid editor in FS-Pile can be used. This is started from the **File :Soil Data Edit** menu command.



The sub-menu is used to select what type of soil data file is to be created/edited. The example below shows an API Soil Profile in the grid editor.



FS-Pile

	Depth m	Int Frict	Shear St	Sub Dens	NDiv
1	0	0	20	9.9	0
2	6.5	0	70	9.9	6
3	9	0	200	10.5	3
4	9	35	0	9.1	3
5	14.5	35	0	9.1	5
6	14.5	0	400	11.1	5
7	19.5	0	400	0.1	4
8	27	0	200	9.2	4
9	33.5	0	300	9.2	6
10	33.5	35	0	9.6	6
11	38	35	0	9.6	5
12	38	0	400	10	5
13	50	0	250	10	10
14					
15					
16					

-o-

5.1 User Defined Soil Profiles API

This is used to define soil properties in terms of defined profiles i.e. as a function of depth. Two files are used to define API pile profiles. On is the primary property file (**.USP**)and the other is an optional extended properties file (**.USPE**). A single file (**.USP**) is used to define the ICP soil profile.

Do not use tabs to delimit fields, only commas or spaces are permitted to be used.

API Primary Soil Profile

The default file extension for the primary file type is **<name>.USP**.

The format of the primary file is;

Depth (m), ϕ (IntFrictionAngle-Deg), ShearStrength (kPa), SubmergedDensity (kN/m3), NDiv

The file has to have the following termination line.

-1, 0, 0, 0, 0

If the FrictionAngle is zero the soil will be assumed to be a cohesive soil and a corresponding value for the Shear Strength must be specified.

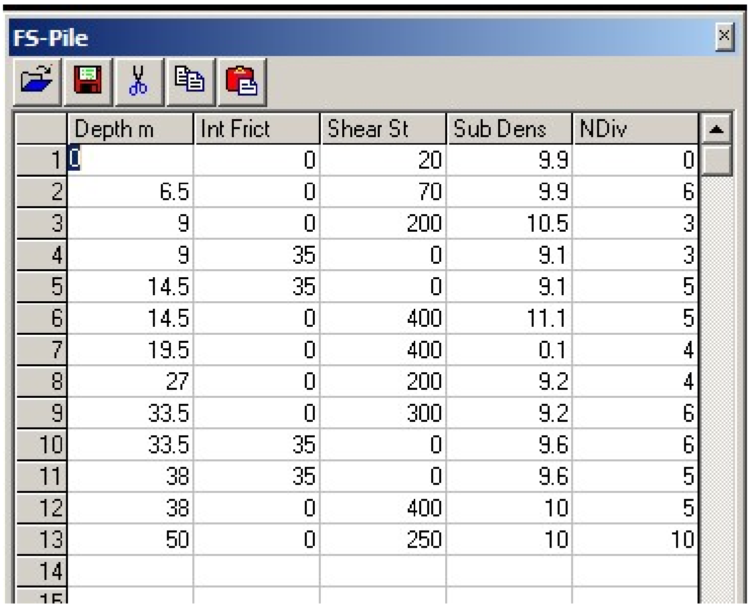
If the Friction Angle is non-zero value, a cohesionless soil is assumed. In this case any corresponding value of Shear Strength will be ignored i.e. taken as zero.

NDiv defines the number elements to be created between each data point.

In layered soils the two soil types must be defined at the same elevation. In the example below this occurs at the 9, 14.5 and 38 elevations.

A typical data file is shown below. This uses commas to delimit the data fields.

```
0, 0, 20, 9.9, 0
6.5, 0, 70, 9.9, 6
9, 0, 200, 10.5, 3
9, 35, 0, 9.1, 3
14.5, 35, 0, 9.1, 5
14.5, 0, 400, 11.1, 5
19.5, 0 , 400 , .1, 4
27, 0, 200, 9.2, 4
33.5, 0, 300, 9.2, 6
33.5 , 35, 0, 9.6, 6
38, 35, 0, 9.6, 5
38 , 0, 400, 10, 5
50 , 0 , 250, 10, 10
-1, 0, 0, 0, 0
```



	Depth m	Int Frict	Shear St	Sub Dens	NDiv
1	0	0	20	9.9	0
2	6.5	0	70	9.9	6
3	9	0	200	10.5	3
4	9	35	0	9.1	3
5	14.5	35	0	9.1	5
6	14.5	0	400	11.1	5
7	19.5	0	400	0.1	4
8	27	0	200	9.2	4
9	33.5	0	300	9.2	6
10	33.5	35	0	9.6	6
11	38	35	0	9.6	5
12	38	0	400	10	5
13	50	0	250	10	10
14					

Specifying Local Scour

In the scour zone the shear strength and soil friction is zero i.e. there is no soil. It would be normal to specify a soil density in the scour zone since local scour will not effect the overburden on the lower sections of the pile. If the soil density is specified as zero in the scour zone the overburden pressure will only start at the bottom of the scour.

Extended Soil Profile

This file optional. If it exists it will be read.

The filename for the extended profile is the primary profile filename with E appended to the end ie **<name>.USPE**. The profile elevations must corrspond exaclty to those of the primary file. If points are defined as zero they will be assigned the global value.

The format of the extended file is

Depth (m), 50%Strain , J-PYConst

A typical data file is shown below. This uses spaces to delimit the data fields.

0	**0 .015**	**.25**
6.5	**0.015**	**.25**
9	**0 .015**	**.25**
9	**0**	**0**
14.5	**0**	**0**
14.5	**0.015**	**.28**
19.5	**0 .015**	**.2**
27	**0 .02**	**.25**
33.5	**0 .02**	**.25**
33.5	**0**	**0**
38	**0**	**0**
38	**0.01**	**.25**
50	**0.01**	**.25**

-o-

5.2 User Defined Soil Profiles

ICP Method Soil Profile
The default file extension for the primary file type is **<name>.USPC**

The format of the primary file is

Depth (m), ϕ, su, δ, qc, Dr, YSR, St, UnitWt, NDiv

Where:

ϕ = Internal Soil friction angle - Used for P-Y curve generation

su = ShearStrength (kPa) - Used for P-Y curve generation

δ = Soil/pile interface friction angle

qc = CPT end restistance (MPa)

Dr = Sand relative density ratio

YSR = Clay Yield Strenght Ratio (apparent OCR)

St = Clay senstivity

UnitWt = Submerged density (kN/m3)

NDiv defines the number elements to be created between each data point

The file has to have the following termination line

-1, 0,0, 0, 0, 0, 0, 0, 0,0

If the YSR is zero the soil will be assumed to be a sand for ICP capacity evaluation.

The API CPT methods are only applicable to cohesionless soils. If these methods are to be used then YSR and St can and should (for clarity) be entered using zero values.For stratified soils the API CPT methods for the clay layers will be implemented using the API method or the API Commentry method.

In layered soils, the interface between the two soil types (sand and clay) is defined by specifying properties twice at the same elevation. In the example below this occurs at the 11.2m elevation.

A typical data file is shown below. This uses spaces to delimit the data fields.

```
0 28 0 23 40 1 0 0 9.7 2
2.1 28 0 23 40 1 0 0 9.7 2
3 28 0 23 40 1 0 0 9.7 2
4 28 0 23 43.3 1 0 0 9.7 2
5 28 0 23 46.67 1 0 0 9.7 2
6 28 0 23 50 1 0 0 9.7 2
7 28 0 23 50 1 0 0 9.7 2
8 28 0 23 50 1 0 0 9.7 2
9.5 28 0 23 50 1 0 0 9.7 2
10.5 28 0 23 50 1 0 0 9.7 2
11.2 28 0 23 50 1 0 0 9.7 2
11.2 0 333 11.5 6 0 5.2 1.65 10.3 2
13.7 0 333 11.5 6 0 4.55 1.525 10.3 2
14 0 333 11.5 6 0 4.472 1.51 10.3 2
15 0 277 11.5 5 0 4.212 1.46 10.3 2
-1 0 0 0 0 0 0 0 0 0
```

FS-Pile

	Depth m	Soil Frict	Shear St	IntFac Frict	qc	Dr	YSR	St	Sub Dens	NDiv
1	0	28	0	23	40	1	0	0	9.7	2
2	2.1	28	0	23	40	1	0	0	9.7	2
3	3	28	0	23	40	1	0	0	9.7	2
4	4	28	0	23	43.3	1	0	0	9.7	2
5	5	28	0	23	46.67	1	0	0	9.7	2
6	6	28	0	23	50	1	0	0	9.7	2
7	7	28	0	23	50	1	0	0	9.7	2
8	8	28	0	23	50	1	0	0	9.7	2
9	9.5	28	0	23	50	1	0	0	9.7	2
10	10.5	28	0	23	50	1	0	0	9.7	2
11	11.2	28	0	23	50	1	0	0	9.7	2
12	11.2	0	333	11.5	6	0	5.2	1.65	10.3	2
13	13.7	0	333	11.5	6	0	4.55	1.525	10.3	2
14	14	0	333	11.5	6	0	4.472	1.51	10.3	2
15	15	0	277	11.5	5	0	4.212	1.46	10.3	2
16										
17										

```
0.00    28  0    23    40     1.0  0     0     9.7   2
2.10    28  0    23    40     1.0  0     0     9.7   2
3.00    28  0    23    40     1.0  0     0     9.7   2
4.00    28  0    23    43.3   1.0  0     0     9.7   2
5.00    28  0    23    46.67  1.0  0     0     9.7   2
6.00    28  0    23    50     1.0  0     0     9.7   2
7.00    28  0    23    50     1.0  0     0     9.7   2
8.00    28  0    23    50     1.0  0     0     9.7   2
9.50    28  0    23    50     1.0  0     0     9.7   2
10.50   28  0    23    50     1.0  0     0     9.7   2
11.20   28  0    23    50     1.0  0     0     9.7   2
11.20   0   333  11.5  6      0    5.2   1.65  10.3  2
13.70   0   333  11.5  6      0    4.55  1.525 10.3  2
14.00   0   333  11.5  6      0    4.472 1.51  10.3  2
15.00   0   277  11.5  5      0    4.212 1.46  10.3  2
-1      0   0    0     0      0     0     0     0    0
```

-o-

5.3 User Defined Soil Stiffness Data

P-Y data can be read directly from a user file to enable the definition of soil stiffness data to be used for subsequent stiffness solution.

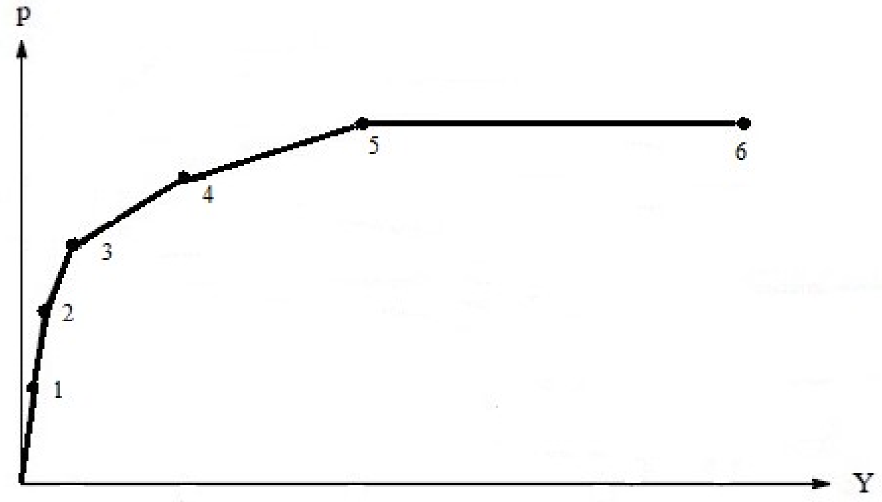
In its simplest mode of operation the number of pile segment divisions are defined and the stiffness curves are interpolated to match the spring locations using equal element lengths. With a stratified soil profile it may be desirable to provide more control over the matching of element lengths and soil springs. It is also possible to align the node locations to match the spring locations i.e. a node for every spring. This is achieved by using data specified in a **<model>.ULPY** file. This file is described at the end of this section.

The default file extension for this file type is **<name>.UPY**.

The units are	P-Y	kN/m & mm	
	T-Z	kPa & mm	[kN/m, mm]
	Q-Z	kPa & mm	[kN, mm]

The diameters (TZDia & QZDia) should only be specified as a non zero value if the units are as specified in [kN/m, mm]. In this case the program will use the diameters to convert kN to the primary stress units i.e. kPa.

It is necessary to always define **6 points** on the curve. If the original data does not contain 6 points then add additional end points. Do not specify the origin point, this is implicit.



Do not use tabs to delimit fields, only commas or spaces are permitted to be used.

The file format is:

P-Y *Comment Line*

NPy

0 *Zero Line not used*

Depth(1), P1, Y1, -------P6, Y6 *(kN/m, mm)*

to

Depth(NPy), P1, Y1, -------P6, Y6

T-Z *Comment Line*

NTz

TZDia *Used only for units conversion*

Depth(1), T1, Z1, -------T6, Z6 *(kPa or kN/m, mm)* to

Depth(NTz), T1, Z1, -------T6, Z6

Q-Z *Comment Line*

NQz

QZDia *Used only for units conversion*

Depth(1), Q1, Z1, -------Q6, Z6 *(kPa or kN, mm)*

to

Depth(NQz), Q1, Z1, -------Q6, Z6

A typical data file is shown below. This uses commas to delimit the data fields

py units in kN/m, mm

8

0

0, 400,17.4, 2,19.1, 3,34.1, 3,915, 3,916, 3,917

0.1, 400,17.4, 2,19.1, 3,34.1, 3,915, 3,916, 3,917

2.9, 2853,17.4, 247,19.1, 317,34.1, 317,915, 317,916, 317,917

4.1, 2853,17.4, 247,19.1, 317,34.1, 317,915, 317,916, 317,917

5.0, 906,32.9, 790,36.2, 790,915, 790,916, 790,917, 790,918

5.4, 906,32.9, 906,915, 906,916, 906,917, 906,918, 906,919

10.5,2703,17.4, 2703,915, 2703,916, 2703,917, 2703,918, 2703,919

21.4,2703,17.4, 2703,915, 2703,916, 2703,917, 2703,918, 2703,919

tz unit in kN/m, mm

12

915

0 ,338,3, 675,6, 1125,10, 1125,15, 1125,18, 1125,20

0.4 ,338,3, 675,6, 1125,10, 1125,15, 1125,18, 1125,20

0.41,503,3,1005,6, 1676,10, 1676,15, 1676,18, 1676,20

3.70,503,3,1005,6, 1676,10, 1676,15, 1676,18, 1676,20

3.71,269,3, 539,6, 898,10, 898,15, 898,18, 898,20

8.45,269,3, 539,6, 898,10, 898,15, 898,18, 898,20

8.46,187,3, 373,6, 373,10, 622,15, 622,18, 622,20

9.85,187,3, 373,6, 373,10, 622,15, 622,18, 622,20

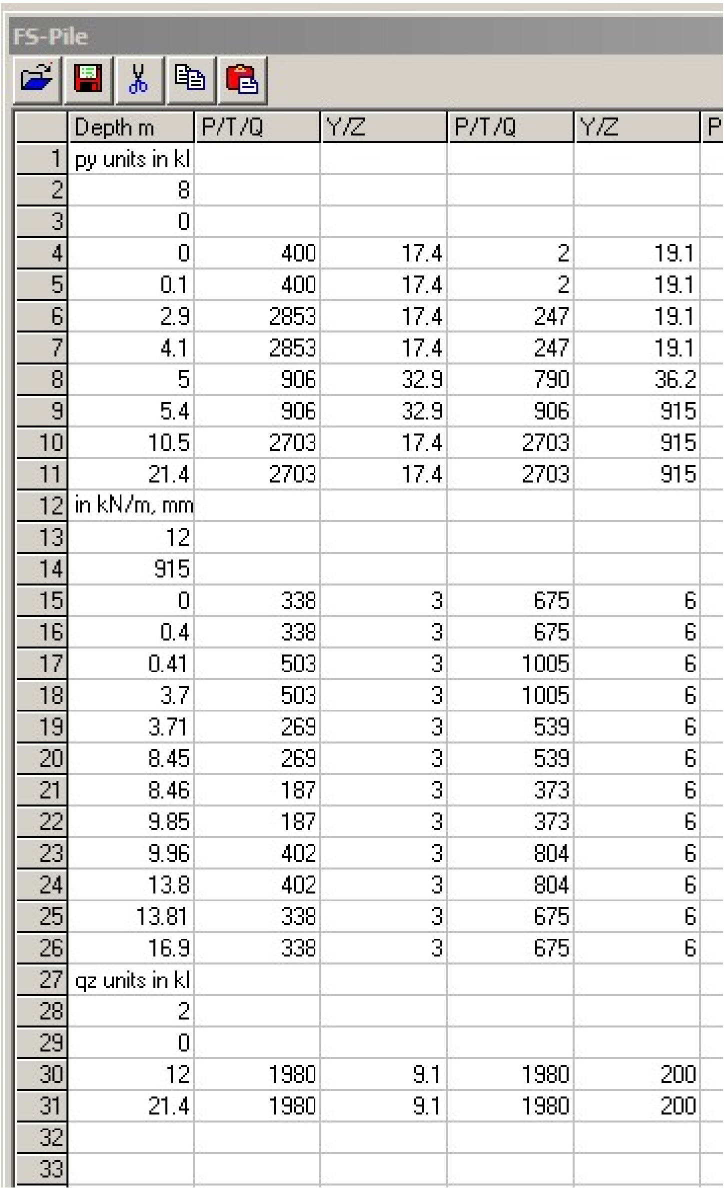
9.96,402,3, 804,6, 1341,10, 1341,15, 1341,18, 1341,20

13.8,402,3, 804,6, 1341,10, 1341,15, 1341,18, 1341,20

13.81,338,3, 675,6, 1125,10, 1125,15, 1125,18, 1125,20

16.9,338,3, 675,6, 1125,10, 1125,15, 1125,18, 1125,20

qz units in kPa, mm



FS-Pile

	Depth m	P/T/Q	Y/Z	P/T/Q	Y/Z	P
1	py units in kl					
2	8					
3	0					
4	0	400	17.4	2	19.1	
5	0.1	400	17.4	2	19.1	
6	2.9	2853	17.4	247	19.1	
7	4.1	2853	17.4	247	19.1	
8	5	906	32.9	790	36.2	
9	5.4	906	32.9	906	915	
10	10.5	2703	17.4	2703	915	
11	21.4	2703	17.4	2703	915	
12	in kN/m, mm					
13	12					
14	915					
15	0	338	3	675	6	
16	0.4	338	3	675	6	
17	0.41	503	3	1005	6	
18	3.7	503	3	1005	6	
19	3.71	269	3	539	6	
20	8.45	269	3	539	6	
21	8.46	187	3	373	6	
22	9.85	187	3	373	6	
23	9.96	402	3	804	6	
24	13.8	402	3	804	6	
25	13.81	338	3	675	6	
26	16.9	338	3	675	6	
27	qz units in kl					
28	2					
29	0					
30	12	1980	9.1	1980	200	
31	21.4	1980	9.1	1980	200	
32						
33						

2

0

12,1980,9.1, 1980,200, 1980,210, 1980,220, 1980,230, 1980,240

21.4,1980,9.1, 1980,200, 1980,210, 1980,220, 1980,230, 1980,240

Defining Non-Uniform Element Sizes

A **<model>.ULPY** file is used to control the meshing of elements. If this file exists it will be used otherwise a prompt for the number of element divisions will appear and the subsequent element length will be NDiv/Pile_Length.

This file has the same format as the USP file and but only the Depth and NDiv values are used.

The last line defines the pile depth. Note that the PY data can and will often extend beyond the pile length, but it should not be shorter.

Two more examples are given below.

The one on the LHS example puts a node at each spring location (assuming the springs are at the same location) and interpolates between 10 and 30 using 10 divisions i.e.a 2m length..

The RHS example uses interpolation and also uses coincident elevation definitions to define boundaries between which interpolation will be undertaken (useful for stratified soil profiles).

Note that the PY data can and will often extend beyond the pile length, but it cannot be longer.

0, 0, 0, 0, 0	**0, 0, 0, 0, 0**
1, 0, 0, 0, 1	**6.5, 0, 0, 0, 6**
2, 0, 0, 0, 1	**9, 0, 0, 0, 3**
3.5, 0, 0, 0, 1	**9, 0, 0, 0, 3**
4.5, 0, 0, 0, 1	**14.5, 0, 0, 0, 5**
6, 0, 0, 0, 1	**14.5, 0, 0, 0, 5**
8, 0 , 0 , 0, 1	**19.5, 0 , 0 , 0, 4**
10, 0, 0, 0, 1	**27, 0, 0, 0, 4**
30,0,0,0,10	**33.5, 0, 0, 0, 6**
-1, 0, 0, 0, 0	**33.5 , 0, 0, 0, 6**
	38, 35, 0, 0, 5
	38 , 0, 0, 0, 5
	50 , 0 , 0, 00, 10
	-1, 0, 0, 0, 0

-o-

5.4 Program Generated Stiffness Data

Two FS-Pile generated file are used during the solution of a pile model.The first is the **UNP** file that defines the non-linea soil spring properties and the second is the **UPN** file, that associates the elements with the non-linear springs.

It is also possible to use Type 7 Couples to represent the soil springs - see Section 5.4.1

UNP File

The format of the UNP file is shown below (note that points 9 to 33 have been deleted) . The first line defines the number of spring points
This file would be used on a model that uses 40 elements in a pile. The end point (41) represents a dummy pile tip element used for end bearing and option tip shear.
Each point represent the soil spring stiffness at the mid point of a pile element. The elevation of this point is also shown.
The first section represents the axial stiffness and the second section the lateral stiffness.

```
 41
 1 -0.2500005  4.309567E-18  9.7536E-04  7.182612E-18  1.88976E-03  1.077392E-17
3.47472E-03  1.29287E-17  0.0048768  1.436522E-17  0.006096  1.149218E-17  0.012192
 2 -0.75  5.74609E-18  9.7536E-04  9.576816E-18  1.88976E-03  1.436522E-17
3.47472E-03  1.723827E-17  0.0048768  1.915363E-17  0.006096  1.532291E-17  0.012192
 3 -1.25  1219.985  0.000635  2439.971  0.00127  3415.959  0.001778  4147.95  0.002159
4879.941  0.00254  4879.941  0.01
 4 -1.75  1707.98  0.000635  3415.96  0.00127  4782.344  0.001778  5807.131  0.002159
6831.919  0.00254  6831.919  0.01
 5 -2.25  2195.974  0.000635  4391.949  0.00127  6148.729  0.001778  7466.313
0.002159  8783.897  0.00254  8783.897  0.01
 6 -2.75  2683.969  0.000635  5367.938  0.00127  7515.113  0.001778  9125.494
0.002159  10735.88  0.00254  10735.88  0.01
 7 -3.25  3171.963  0.000635  6343.927  0.00127  8881.498  0.001778  10784.67
0.002159  12687.85  0.00254  12687.85  0.01
 8 -3.75  3659.958  0.000635  7319.916  0.00127  10247.88  0.001778  12443.86
0.002159  14639.83  0.00254  14639.83  0.01
 -
 -
 34 -16.75  21016.54  9.7536E-04  35027.57  1.88976E-03  52541.36  3.47472E-03
63049.63  0.0048768  70055.15  0.006096  56044.12  0.012192
 35 -17.25  21567.84  9.7536E-04  35946.39  1.88976E-03  53919.59  3.47472E-03
64703.51  0.0048768  71892.79  0.006096  57514.23  0.012192
 36 -17.75  22118.21  9.7536E-04  36863.68  1.88976E-03  55295.52  3.47472E-03
66354.63  0.0048768  73727.36  0.006096  58981.89  0.012192
 37 -18.25  22667.67  9.7536E-04  37779.45  1.88976E-03  56669.18  3.47472E-03
68003.01  0.0048768  75558.91  0.006096  60447.13  0.012192
 38 -18.75  23216.23  9.7536E-04  38693.72  1.88976E-03  58040.58  3.47472E-03
69648.69  0.0048768  77387.44  0.006096  61909.95  0.012192
 39 -19.25  23763.89  9.7536E-04  39606.48  1.88976E-03  59409.73  3.47472E-03
71291.67  0.0048768  79212.97  0.006096  63370.38  0.012192
 40 -19.75  24310.66  9.7536E-04  40517.77  1.88976E-03  60776.65  3.47472E-03
72931.98  0.0048768  81035.54  0.006096  64828.43  0.012192
 41 -20  258332.7  0.0012192  516665.3  7.924801E-03  774997.9  0.0256032  929997.6
0.0445008  1033331  0.06096  1033331  6.705601E-02
 1 -0.2500005  1.273378E-17  0.001524  1.836462E-17  0.004572  2.7432E-17  0.01524
3.956234E-17  0.04572  5.48602E-17  0.12192  5.4864E-17  0.27432
 2 -0.75  1.273378E-17  0.001524  1.836462E-17  0.004572  2.7432E-17  0.01524
3.956234E-17  0.04572  5.48602E-17  0.12192  5.4864E-17  0.27432
 3 -1.25  18933.93  0.002286  32010.2  0.005715  35509.8  0.01143  35754.02  0.018288
```

```
35763.41  0.02286  35764.77  0.1524
 4 -1.75  26093.5  0.002286  42232.19  0.005715  45507.73  0.01143  45643.17  0.018288
45645.8  0.02286  45646.02  0.1524
 5 -2.25  34279.64  0.002286  58470.16  0.005715  64859.36  0.01143  65257.43
0.018288  65269.97  0.02286  65271.48  0.1524
 6 -2.75  42950.36  0.002286  78184.16  0.005715  90523.61  0.01143  91652.18
0.018288  91704.86  0.02286  91713.42  0.1524
 7 -3.25  51641.71  0.002286  98995.74  0.005715  119751.9  0.01143  122341  0.018288
122506  0.02286  122540.5  0.1524
 8 -3.75  60329.93  0.002286  120511.9  0.005715  152132.5  0.01143  157226.6
0.018288  157644.4  0.02286  157752.6  0.1524
 -
 -
 34 -16.75  53985.94  0.001524  77858.35  0.004572  116300.3  0.01524  167727.9
0.04572  232584.4  0.12192  232600.5  0.27432
 35 -17.25  55312.38  0.001524  79771.33  0.004572  119157.8  0.01524  171848.9
0.04572  238299  0.12192  238315.5  0.27432
 36 -17.75  56638.82  0.001524  81684.31  0.004572  122015.3  0.01524  175970  0.04572
244013.6  0.12192  244030.5  0.27432
 37 -18.25  57965.25  0.001524  83597.29  0.004572  124872.8  0.01524  180091.1
0.04572  249728.2  0.12192  249745.5  0.27432
 38 -18.75  59291.68  0.001524  85510.27  0.004572  127730.3  0.01524  184212.2
0.04572  255442.8  0.12192  255460.5  0.27432
 39 -19.25  60618.12  0.001524  87423.26  0.004572  130587.8  0.01524  188333.3
0.04572  261157.4  0.12192  261175.5  0.27432
 40 -19.75  61944.55  0.001524  89336.23  0.004572  133445.3  0.01524  192454.3
0.04572  266872  0.12192  266890.5  0.27432
 41 -20 0 0.001524 0 0.004572 0 0.01524 0 0.04572 0 0.12192 0 0.27432
 1
 1
```

The UPN File

The UPN file associates the soil spring stiffness with a pile element. The element must be a Type 6 beam element.

The extract from a UPN file shown below shows the format used, 100 is the element and 41 is the spring number etc

```
 100  41
 101  40
 102  39
 103  38
 104  37
 105  36
 106  35
 107  34
 108  33
 109  32
 110  31
 111  30
 112  29
 113  28
 114  27
```

-o-

5.4.1 Using Non-Linear Couples

Normally soil springs are associated with the pile elements and are essentially an internal attribute used by the solvers. An alternative approach is to use Type 7 couples connected to the pile element nodes.

This approach would not often be required. It would be necessary if seismic time history solution is a requirement i.e. displacements applied to ground springs or in cases where more complex soil model were required to be used.

When this approach is adopted the Type 7 couples and their associated RC constants are included in the UM file. The RC properties are contained in a UMP_RC file which must be interpreted when the UM file is interpreted.

When this method is used the UPN file must not exist otherwise the spring stiffness will be doubled (the program will delete the UPN file when a pile segment is generated). Deletion of the UNP file is optional.

In this method each pile node is copied 1 m in the x -direction. A Node to Node Type 7 couple is then connected between the pile node and the copied node for element using the element as the reference coordinate for the couples. This results in two couples being attached node connecting two pipe elements and one at the ends.

The copied node has to be restrained in all degrees of freedom and this is required to be done by the user.

The node increment will be the double of that required for the element increment

The starting Couple and the Couple Property Labels are defined prior to pile segment generation using the following entry prompt.

The Couple start is define using an offset of the starting element label. In this example the Couple start label would be 150 if the starting element label was 350.
The Couple Property start code would be 20.

The RC property table contains the non-linear soil stiffness data. This data will always start at RC code entry 11 therefore ensure that any existing model data does not use these entries.

Note that the pile elements need not be a Type 6 beams when using Couples.

The only addition consideration when using couples are:

- Ensure the **Label Increment** is large enough for additional nodes > 2 x Number of Element in Pile Segment.
- Be ready to define the **Couple Label Offset** and **Property Code Start.**
- In addition to the **UM** files interpret the **UMP_RC** file.
- Fully restrain the the couple ground nodes.
- Ensure that RC 11 codes and upwards are not used in the original model.

-o-

6.0 Program Files Used in Pile Analysis

In addition to the pile option files used by FS-Pile, the following files are also used

File Ext	**Description**
Definition Data	
The following data files will be archived with the model (U files)	
.UPL	Basic Pile Model Data
.USP	User Defined Soil Profile for Standard API Methods
.USPE	Extended API related data (Optional)
.USPC	User Defined Soil Profile for CPT Methods
.UPY	User Defined Soil Stiffness Data
.UPLY	User Defined Element Mesh Control - Optional use with a UPY defined stiffness.
.UNP	Non-linear spring data used by solvers (Program generated)
.UPN	Non-Linear Spring Association solvers. (Program generated)
.UM'n'	Model command file (Program generated)
.UMP_RC	Model command file RC curves (Program generated)
Defintion Data - Pile Driving	
.UPDRV	Pile Driving & Stiffness Data (Program generated)
Results Data - Basic Pile Properties	
.DPR	Pile Performance & Stiffness Data
.DSP	Soil Profile for Linear Soils
	(program generated data associated with linear soil data)

Non-Linear Analysis Results form FS-Pile

.DA"n" Axial Force/Deflection Plot File

.DL"n" Lateral Force/Deflection Plot File

.DR"n" Moment/Deflection Plot File

Deleting Piles from a Model

If pile elements are required deleted from a model then the the UPN file must be also be deleted. This prevents future elements with the same label form be assigned pile spring properties.

-o-

7.0 Pile Driving Analysis

The Pile Pile Driving menu option enables pile driving analysis by Smith's Wave Equation to be undertaken. (Ref. Smith, I.M. 'Pile driving by the Wave Equation', ASCE, Paper No 3306, Vol 6, 1962, Part 1) .

Pile Driving Model

The driving analysis does not require an FS2000 model to be created, an independent driving model is used based on the same model that would be created when interpreting the UM files create by FS-Pile.

The solution is based on a finite difference approach in which the pile is divided into segments in an identical manner to that used in the non-linear pile analysis. The components of the driving system are represented by a series of masses connected by springs. The soil profile should be defined such that the resulting pile segments are of a similar size. They need not be identical.

Soil Properties

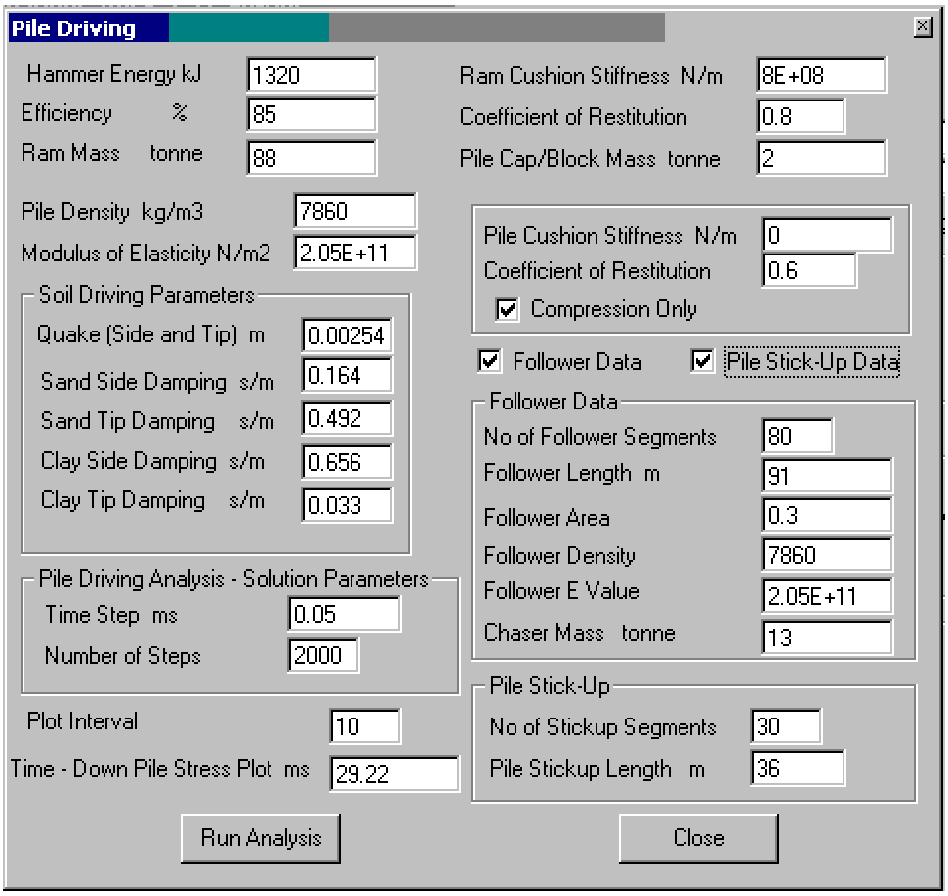
The resistance to driving is based on the skin friction and the end bearing from the defined soil profile and is interpreted the same as in conventional static pile analysis. A driving resistance to take account of reduced dynamic resistance can be conveniently achieved by using an additional defined soil profile file e.g. .USPSRD.

The effect of driving at different depths is obtained by simply changing the pile length in pile the pile definition form.

A typical pile configuration is shown in Section 7.1 along with typical underwater pile hammer data (IHC Hammers).

Driving Data Input

The first menu option will make the following form visible. The other menu options enable the pile driving results to be plotted.



Apart from the Solution Parameters the data input should be self explanatory. Input Data on the RHS of the form is optional data.

The **Ram Cushion Stiffness** is for the an optional cushion that is directly below the ram.

The **Pile Cushion Stiffness** is for the an optional cushion that is directly below the pile cap or the follower chaser if present.

If the **Follower Data** and **Pile Stick-Up** Options are active the form can be used to define the optional data. Usually only present when concrete pile are being driven.

If the **No of Stickup Segments** is defined as non zero the **Stickup Length** must be specified.

If the **No of Follower Segments** is defined as non zero the remaining follower data must be specified. Setting the **No of Follower Segments** equal to zero eliminates the follower**.**

The **Time Step** defines the time interval used for the finite difference solution. The smaller the value the less likely numerical instabilities will occur. A time step of .025 ms will generally give good results but may require to be smaller (0.005). The size of the time step is related to the size of the pile elements.

The **Number of Step** effectively defined the solution time. This should be large enough so that time at which the pile tip rebounds is included. This depend upon the length of the pile. The wave travels down a steel pile at about 5000 m/s so this give some indication of the solution times. It is essential to Plot the Pile Tip Displacement to ensure that the solution time is sufficient. The solutions obtained at times beyond the maximum tip displacement are not significant.

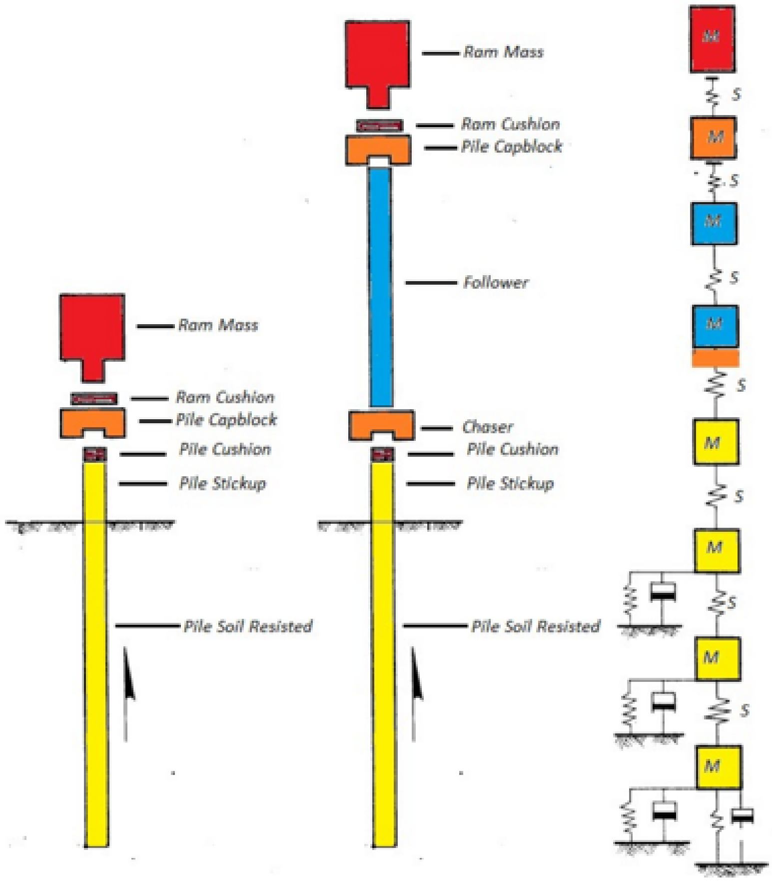
Important - It is essential that the size of the time step does not significantly change the analysis results for the the same total analysis time. If it does, the solution has not converged and a smaller time step should be used. Always plot the Tip Displacement to ensure the correctness of the solution.

The **Plot Interval** is used to define the time step plot interval. The maximum number of plot points is 1000 so the plot interval should be set accordingly.

The **Time - Down Pile Stress Plot** defined the time at which the pile stress plot is created. The stress plot includes both the follower and pile stickup if present

-o-

7.1 Pile Configurations



HAMMER S SERIES		S-30	S-40	S-70	S-90	S-120	S-150	S-200	S-280	S-500	S-600	S-800	S-900	S-1200	S-1400	S-1800
OPERATIONAL DATA																
Max. blow energy on the pile	kNm	30	40	70	90	120	150	200	280	500	600	800	900	1200	1400	1800
Min. blow energy on the pile	kNm	3	4	7	9	12	15	20	28	50	60	80	90	120	140	180
Blowrate at max. blow energy (1)	bl/min	65	65	50	46	48	44	45	45	45	42	38	38	38	40	35
WEIGHTS																
Ram	ton	1.6	2.2	3.5	4.5	6.2	7.5	10	13.6	25	30	40	43	60	69	90
Hammer with ram in air (2,3)	ton	3.9	4.7	8.3	9.7	14.3	16.2	25.8	30.5	57.5	64	83	120	140	148	210
DIMENSIONS																
Length hammer (4)	mm	6100	6850	7400	8055	8166	8900	9095	10390	11943	12715	14610	12795	14297	16090	16510

-o-