

# FS2000

***ISO 19902/API RP2A LFRD Member Design***

***Advanced Structural Analysis  
for Windows  
(c) A.E.S. Ltd 1988,2016***

***Email: [support@aes-uk.com](mailto:support@aes-uk.com)***

# Table of Contents

1.0	Introduction	3
2.0	Creating Models in FS2000	4
2.1	Units	5
2.2	Creating a Processed Results File	6
2.3	Property Codes	7
2.4	Structural Design Parameters	8
3.0	Operation of the CodeChecker	9
3.1	Operation from FS2000	10
3.2	CodeChecker - Interactive Operation	12
3.3	Property Tabel Lists	13
4.0	Modification of Element Properties	14
4.1	Buckling Parameters	16
4.2	Moment Coefficients	17
4.3	Property Code Data	18
4.4	Buckling Parameters - Graphical Definition	19
5.0	Technical Specification (ISO 19902)	20
6.0	Technical Specification (API RP2A LRFD)	21
APPENDIX A - Geometric Property Libraries		22
APPENDIX B - Data Files		26
APPENDIX C - Batch Operation		27

## **1.0 Introduction**

FS\_ISO is an interactive program module that interfaces with FS2000 to provide cylindrical design checks in accordance with the code requirements of BS EN 19902:2007, Petroleum and natural gas industries - Fixed steel offshore structures.

FS-APILRFD is an interactive program module the interfaces with FS2000 to provide cylindrical design checks in accordance with the code requirements of API RPA-LRFD, Recommended Practice for Planning, Designing and Constructing Fixed Offshore Platforms - Load and Resistance Factor Design.

The units of the code checker are consistent with the standard SI structural property tables or US structural property tables. All output clearly defines the units employed.

-0-

## **2.0 Creating Models in FS2000**

This section describes the few requirements and considerations necessary when creating models that are to be code checked.

Most design parameters used in a design check are obtained the basic model data defined during model creation within FS2000. It is essential to confirm that the certain design parameters such as buckling lengths which can be are appropriate. Section 2.4 list the design parameters used in the design check.

-0-

## 2.1 Units

AISCCodeCheck reads the inputs and result files of models created and analysed by FS2000. To ensure units compatibility it is essential that the model be created using the either the **SI-Unit** or **USA-Unit** system in FS2000.

The basic unit requirements for the two systems are defined below.

### **SI System**

Length	metres (m)
Force	Newton (N)
Moment	Newton-metre (Nm)
Stress	N/m <sup>2</sup> (typically for steel E=205E9 Grade 50 Yield=345E6)

### **USA System**

Length	inches (ins)
Force	pound (Lbs)
Moment	pound-inch (Lb-ins)
Stress	Lbs/ins <sup>2</sup> (psi) (typically for steel E=30E6 Grade 50 Yield=50E3)

-0-

## **2.2 Creating a Processed Results File**

Before the codechecker can be run the analysis case results must be processed in the Post-Processor. The post processor converts raw results cases into processed results cases. It is also used to combined multiple results cases in to single processed results using pre-defined load case combinations.

-O-

## 2.3 Property Codes

The main model parameters that are required to be identified are :

- 1 Section properties of the elements, and
- 2 Element steel grades

For a section property to be checked it must exist in a section model library ie it must have been selected from a property library when creating the model in FS2000.

The section properties are identified by the program by reading the section library and comparing the Designation. When a match is found the property code of the model are assigned to the appropriate section. The exception to the previous for property identification is for the identification of circular hollow sections (CHS) and angles. With CHS's the properties are identified by outside diameter and wall thickness.

When the model file data has been loaded the property codes may be reviewed to ensure that property codes were correctly identified. This is described in section 3.0.

### **Angle Sections**

FS2000 has two types of angle sections.

*A Type* angles - These have no bending stiffness, to be used only as struts or ties.

*L Type* angles - These are general beam elements with bending capability

In the case of A Type angles the program will only assess load capacity in terms of axial loading. To ensure that this assumption is maintained during the analysis the element property code should be assigned very low I (or moment releases) values to ensure that moment is not carried by the angle. Zero I values are permissible in FS2000 providing mechanisms are not formed. When A Type angles are selected from the standard libraries of FS2000 low I values are assigned.

-0-

The design parameters used in a design check are obtained the basic model data defined during model creation within FS2000. Modification of these parameters is described in Section4.

#### Tensile Strength

**Area Ratio** Defines the effective area (tensile capacities).

#### Column Buckling

**Eff.lx** Effective Length for compressive buckling about the local xx axis.

**Kx** Effective length for xx Comp buckling

**Eff.Ly** Effective Length for compressive buckling about the local yy axis.

**Ky** Effective length for yy Comp buckling

#### Lateral-Torsional /Buckling

**Llb** Unsupported Length of Compression Flange

**CMx** *Moment reduction factor*

**CMy** *Moment reduction factor*

**Cb** *Lateral buckling parameter*

#### Plate Girder Design

##### **Stiffener spacing**

#### Buckling Lengths

By default the length of the element is assigned to the effective length values above. Often these default lengths will be too short since nodes may have been introduced that reduce the element length.

Using a length that is too short may be **UN-SAFE**.

IT IS ESSENTIAL THAT THESE VALUES REFLECT THE EFFECTIVE LENGTHS IN THE STRUCTURE. USING A LENGTH THAT IS TOO SHORT WILL PRODUCE ALLOWABLE LOADS ABOVE THE ACTUAL CODE ALLOWABLES

#### Moment Factors

The data shown in *italics* relate to factors that dependent upon the moment distribution in the element. In most case the default values will be conservative but the user should check their applicability.

-0-



### **3.0 Operation of the CodeChecker**

The codechecker is started from the Design Menu of the Output/Results TASK in FS2000. When started from here there are two basic modes of operation available. The selection of the mode is by option buttons in the CodeCheck data entry window.

- Create Output - Direct Mode
- Interactive - Interactive Mode

The CodeChecker can be run from FS2000 in direct mode using either the Summary Output or Full Output options. In either mode the output results are created without any further interaction with the user. If the codechecker is run in Batch mode it the same as Direct Mode.

The alternative is to run the CodeChecker in Interactive mode. In this mode the user has more control over what elements are to be checked. Single elements can be checked and the basic design parameters may be changed so as to provide a 'what if' design environment.

The following outline a basic procedure for undertaking design checks.

- Select the Design Parameters TASK and define the buckling lengths for all element whose element lengths do not reflect the true design lengths. This could be done in the code checker but it far more convenient to define within the FS2000 graphical environment.
- Create a results case in FS2000.
- From the Design menu in the Output/Result TASK in FS2000 select the codechecker.
- Run the codechecker initially in interactive mode. This will enable the user to check that all data is correctly interpreted.
- When the Member Design Check is visible click the [Property Table List](#) and ensure that all necessary properties have been correctly identified .
- Click the Selective Element Report and then design check all elements .
- The view window will now show a summary report.

For subsequent design checks run the codechecker in direct mode either from FS2000 in interactive model or in a Batch file.

#### **Reviewing the Output from the Design Checks**

The most convenient way to review the output from the code check is to plot the design utilisation ratios (UR Plot). Search for UR in the FS2000 Help index for further information on UR plots.

The text out from the codechecker can be view one of the following.

- Menu Command - Data:View/Print Report Data:Member Design Results
- Menu Command - Report Collation Data Select
- The design form from the Design menu in the Output/Result TASK in FS2000

-0-

### 3.1 Operation from FS2000

The program is started from the Design Menu of the Report TASK in FS2000. When this command is selected the following input form will appear.

**Results Case** is used to define the Processed Results Case to be code checked. The **Browse** button is used to select from a list.

A range of cases may be processed by defining a range i.e. 1-9 will process results cases between 1 and 9. If a case does not exist a warning will be given and the process will continue.

The **Sub-Case Output** option enables multiple formatted output files for the same Results Case to be created. This may be used if it is desirable to create separate Results files for the different output category options and various sort options available. If the option is checked then the file created will have the file name <Model>.Grp.\* where Grp is the name entered in the Sub-Case description box.

The **Water Depth** is used to define the depth of water of the the combined hydrostatic loading checks. This has two modes of operation depending whether the external pressure is define explicitly in a load case or is not ([See Section 4.0](#)). The length between stiffening rings is based on the **Llb** length parameter which defaults to the element length

**Summary Output** option produces an output listing that shows only the design unity ratio.

**Full Output** option produces an output listing that shows the actual and allowable loading.

**Interactive Mode** option activates Interactive Operation of the CodeChecker

**Number of Locations on Span** defines the number of points along an element at which the code checks are to be carried out. At each point the loading will be output (Full Output listing). Up to 21 points along the length of the element may be specified. If the **MaxUR SpanChk** is active the element will be check at 21 points on the span but only the ends and the mid point with the maximum UR will be listed. When this is active the number of locations will always be set to 3.

**Stress Ratio Limit** is used to restrict the output to the elements whose maximum unity ratios are greater than that specified. The default value is zero. This facility is extremely useful since it reduces the output from the program and identifies critical elements more quickly.

The **Execute** button is used to start the program with the currently shown settings.

The **Batch** button converts the set options to command line switches and appends the AISC command line to the .BRM batch run file.

The **View** button loads the results case file view form.

### Groups

The **Group SET** box is used to define the group SET to be loaded. If a SET is loaded then all node and

element labels will be accompanied by their respective group attribute. If this field is left blank or contains the number of a non-existent group then only the basic node and element numbers will be used for reference in the lists.

The **By Label (All)** option will output all entities (nodes and elements) in ascending label order.

The **By Group Only (to Limit)** option will output entities in ascending Group order. Entities not assigned to groups or entities assigned to Groups greater than defined by the **Group Limit/Restriction** box will not be output. This is a restricted process option.

The **By Group(to Limit) then Label option** will output entities in ascending Group order. Entities not assigned to groups or entities assigned to Groups greater than defined by the **Group Limit/Restriction** will be output in label order following the sorted groups. All data is processed with this option.

The **Restrict to One Group** option is used to restrict entities to only those entities with the same group number as defined by the **Group Limit/Restriction** box (zero value indicates that all data will be shown). This is a restricted process option.

**Important Note** Stress ratio data created by this module will be limited to the data processed. If restricted process options are used then any Stress Ratio plots or Stress Ratio sorts which use the same results cases will be limited to the processed data. The plot or listed output will indicate if the output is from a restricted process e.g. Von-Mises Restricted.

-0-

### 3.2 CodeChecker - Interactive Operation

When the Interactive option is selected the program will be started in interactive mode. In this mode the module runs as a separate windows application to FS2000 and there is no further interaction with FS2000.

When the module starts the following main CodeCheck form will appear.

The screenshot shows a software window titled "ISO 19902:2007 Member Design Rel 8-1-13". Inside the window, there are several input fields and buttons. At the top left, there is a "Results Case" field with a "Browse" button next to it. Below this is a "Groups ID Group SET" field, also with a "Browse" button. To the right of these is a "Help" button. In the center, there are two large buttons: "Property Table List" and "Element Properties". Below these, there are two input fields: "No of Points on Span" (with the value 3) and "Utilisation Ratio Output Limit" (with the value 0.4). To the right of these fields are two checkboxes: "Max UR Span Check" and "Bacoff Forces". At the bottom, there is a wide button labeled "Selective Element Report" and a smaller button labeled "Close Code Check".

Apart from the **Selective Element Report** and the **Element Properties** buttons the input options of the form are identical to those of the [FS2000 form](#) described in the previous section.

The **Property Table List** will show [property lists](#) that show if the geometric and material property code have been correctly interpreted.

The **Selective Element Report** button is used to select individual or sets of elements for selective codecheck. The **ID Group SET** is used to select elements by group. If an individual element is specified then a more detailed output will be given. If multiple elements are selected a summary output will be given. The output from this mode is listed on the screen in a list viewer. It may also be printed directly from this viewer.

When the option is selected a Selection form will become visible. This selection form is used to define the element(s) to be checked

The **Element Properties** button is used to view and define elements properties. Re-defining properties enables the user to design the element for the applied loading. The re-definition of section properties is described in [Section 4.0](#).

The **Bacoff Forces** check box is used input fore end and aft end offsets. These offsets are used to 'back off' forces and moments along the span. This gives the user the facility to take into account span "face to face" load levels.

-0-

### 3.3 Property Table Lists

The following table is used to check that property code data is correctly interpreted.

Geometric & Material Properties							
Code	Designation	Welded	Area Ratio	Girder Stiff Spacing	Yield Strength		
1	CHS 324.0	25.4	N	0.000	0.0	450.0	
2	CHS 219.0	25.4	N	0.000	0.0	355.0	
3	CHS 141.0	12.7	N	0.000	0.0	345.0	
4	PIP 0.1	0.0	N	0.000	0.0	420.0	
5	CHS 457.0	12.7	N	0.000	0.0	358.5	
6	PIP 0.4	0.0	N	0.000	0.0	450.0	
7	CHS 1067.0	25.4	N	0.000	0.0	358.5	
8	PIP 0.5	0.0	N	0.000	0.0	415.0	
9	RHS 250150125		N	0.000	0.0	0.0	
10	CHS 168.3	14.3	N	0.000	0.0	0.0	
11	CHS 60.3	6.4	N	0.000	0.0	355.0	
12	CHS 60.3	4.8	N	0.000	0.0		
13	CHS 60.3	6.4	N	0.000	0.0		
14	CHS 33.4	3.4	N	0.000	0.0		

The table shows both Geometric and Material Property codes in the same list. In the above list there are only 11 Material property codes in the model ie above 11 the Yield Strength field is blank.

Geometric property data not recognized by the codechecker will be indicated by ??? for the property Designation.

Material property data not recognized by the codechecker will be indicated by 0 for the property Yield Strength.

-0-

## 4.0 Modification of Element Properties

When the Element Properties button is pressed the Member Data form shown below will appear. This form is used to define data relating to the code checking of the element.

The Buckling Parameter data section refers to geometric related parameters relating to column and lateral buckling. See [Section 4.1](#)

The Moment Coefficient data section refers to moment distribution related parameters relating to column and lateral buckling. See [Section 4.2](#)

The Property Code data section relates to data defined by Geometric Property Code and Material Property Code reference and is described in [Section 4.3](#)

**Member Data**

**Buckling Parameters**

Elem	Eff Lx	Kx	Eff Ly	Ky	Lib
158	7.5	1	1.875	1	1.875

Enter Modify Save View List

**Moment Coefficients**

Elem	Cm-x	Cm-y	Cb
158	0.85	0.85	1

Enter Modify Save Get View List

**Property Code Data**

Elem	Designation	Yield
158	G5 M1 UB 45719174	345

Girder Stiff Space 0 Area Ratio 1

Enter ☐ CHS (Pipe) ☐ Welded View List

**Hydrostatic Collapse**

Water Depth (Above Model Origin)

0

Close

### Hydro Static Collapse

This parameter is also defined as a primary input parameter i.e batch command line.

In this codecheck the water depth has two modes of operation depending whether the external pressure is defined explicitly in a load case or is not.

- Explicit External Pressure Definition

If the external pressure is defined explicitly in a load case or a load case generated from FS-Wave the **Water Depth** parameter is interpreted as a load factor (range is 0.9 to 1.3) and is applied to the pressure from the factored result case. The value of this should be adjusted so the that the correct load factor is applied to the pressure e.g. 1.3/1.35 would be used if a factor of 1.35 was applied to the load case and 1.3 was required for the Hydrostatic pressure load factors.

- Non-Explicit External Pressure Definition

The **Water Depth** is used to define the depth of water of the combined hydrostatic loading checks. It a non zero value is entered this will be interpreted as depth of water relative to the model Y origin. A negative value would imply that the surface elevation is below the origin. In this mode of operation

(Non-Explicit) the Hydrostatic pressure load factor is always taken as 1.3.

The length between stiffening rings is based on the **L<sub>ib</sub>** length parameter which defaults to the element length.

-0-

## 4.1 Buckling Parameters

The following data boxes are used to enter data relating to member buckling.

Elem	Eff Lx	Kx	Eff Ly	Ky	Llb
1	1	1	1	1	1

Enter    Modify    Save    View List

**Eff.Lx** Effective Length for compressive buckling about the local xx axis.

**Kx** Effective length for xx Comp buckling

**Eff.Ly** Effective Length for compressive buckling about the local yy axis.

**Ky** Effective length for yy Comp buckling

**Llb** Length between ring stiffeners for hydro buckling checks

### UNITS

SI-Units length in metres

USA-Units length in inches

-Also applies to Depth for hydro checks

By default the length of the element is assigned to the effective length values above. Very often these default lengths will be too short since nodes may have been introduced that reduce the element length.

Using a length that is too short may be **UN-SAFE**.

IT IS ESSENTIAL THAT THESE VALUES REFLECT THE EFFECTIVE LENGTHS IN THE STRUCTURE. USING A LENGTH THAT IS TOO SHORT WILL PRODUCE ALLOWABLE LOADS ABOVE THE ACTUAL CODE ALLOWABLES

The **Enter** button is used to enter the data displayed in the data boxes to the element number displayed in the **Elem** box.

The **Modify** button is used to copy the data displayed in the data boxes to other elements. Data is copied to other elements by defining an element label range or an element group. The main CodeCheck form should be used to load the appropriate Group SET.

The **Save** button is used to save all current element buckling data entered to the definition file (.ELN file). This is a formatted text file which should be included in any analysis report. If one exists it will always be loaded when the code check module is started. The file may be edited. The instruction command for the data is ME. If a line is entered all parameters must be present (see Appendix B).

The **ViewList** button is used to display the current buckling data entries for all elements. Elements may be selected for data entry from this form. The Update button on the form is used to update the list if one or more elements are modified.

-0-



## 4.2 Moment Coefficients

The following data boxes are used to enter data relating to member moment distributions.

Moment Coefficients			
Elem	Cm-x	Cm-y	Cb
158	0.85	0.85	1
<input type="button" value="Enter"/> <input type="button" value="Modify"/> <input type="button" value="Save"/> <input type="button" value="Get"/> <input type="button" value="View List"/>			

- CMx**      Moment reduction factor
- CMy**      Moment reduction factor
- Cb**        Lateral buckling parameter

The **Enter**, **Modify** and **ViewList** buttons operate in the same manner as those for the buckling parameters.

The **Save** button is used to save the elements, and only those, whose parameters have been modified. The parameters are save to a '.^m' binary file.. The 'm' is the same number as the results case as these parameters are load case dependent.

The **Get** button is used to retrieve a previously saved moment factor file. Always ensure that the retrieved case is related to the current load case file.

The '.^m ' files will not be loaded during command line operation. These parameters can only be used during interactive operation.

-0-

### 4.3 Property Code Data

This section of the Data form is used to modify parameters relating to the property codes of the elements.

The **ViewList** button is used to show the Geometric and Material properties associated with the property codes of the model. Geometric property data not recognised by the code checker will be indicated by ??? for the property Designation.

The element property codes are identified in the box to the right of the **Elem** box. These entries cannot be changed.

The **Browse** button may be used to change the geometric properties associated with a property code. The main use of this facility is to check the effect of different section types in the design. Changes here are temporary they cannot be saved and do not effect the original model. For entries to be effective the Enter button must be used. Use the **ViewList** to check the entries.

The **Yield** data box is used to re-define the design stress of the material code. The main use of this facility is to check the effect of material strength in the design. Changes here are temporary. They cannot be saved and do not effect the model.

The **Girder Stiff Space** box is used to define stiffener spacing for vertical web stiffeners in plate girders. The spacing length is entered in mm.

The **Area Ratio** box is used to define the  $A_e/A_g$  ratio.  $A_e$  is the effective area used to establish allowable tensile load capacities.

The **Welded** check box is used to identify sections as being of welded construction. Sections built up by welding are identified by preceding the designation with a "-" sign. A property code with a designation PLT-20010010 would assumed to be a welded section. The -ve sign may be included in the section property libraries.

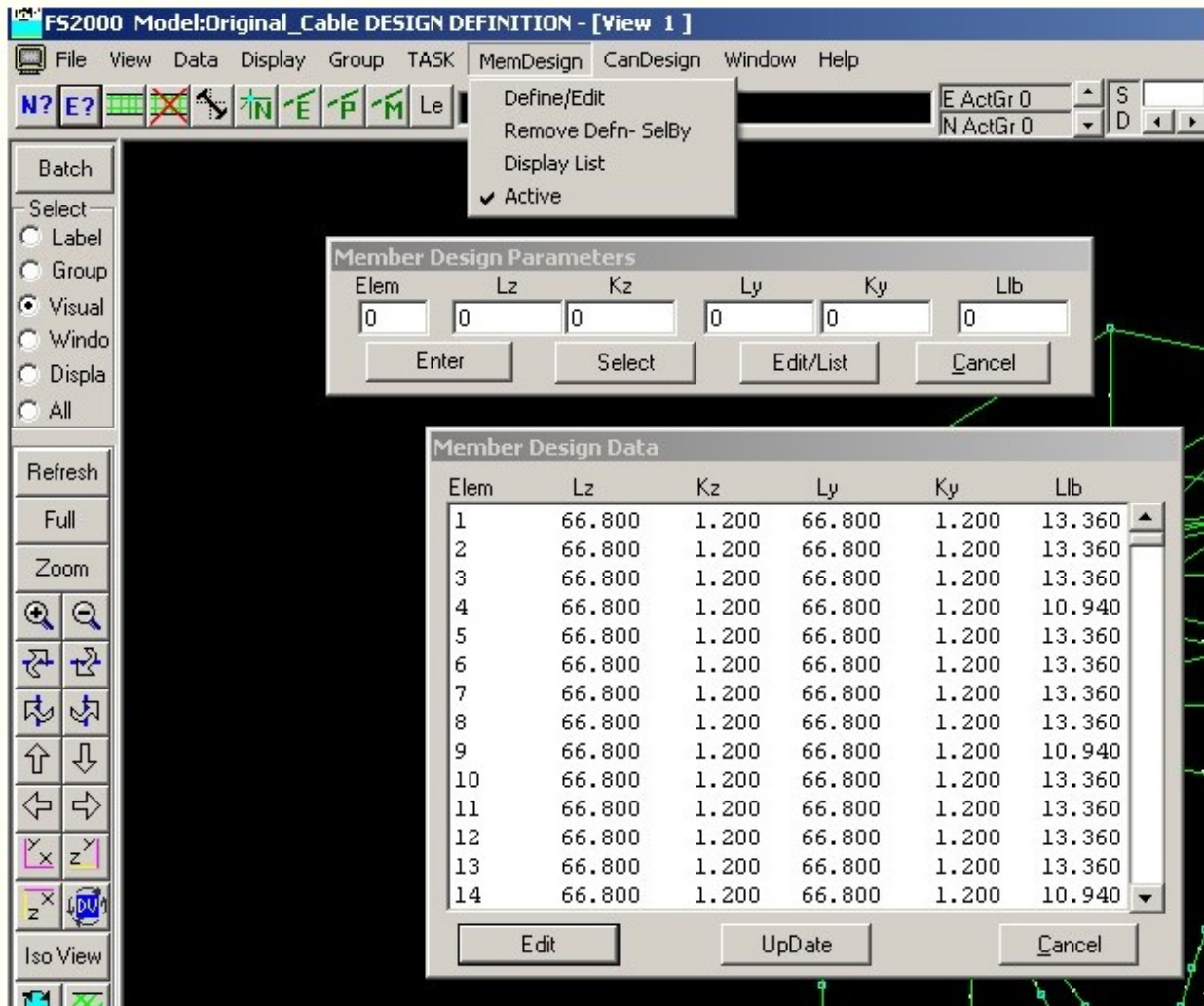
The **CHS** check box is used to change the property code to that for a pipe. When checked, input boxes for the OD and wall thickness of a tube will appear. The Enter button is used to enter the values displayed in the boxes. Use the **ViewList** to check the entries.

-O-

## 4.4 Buckling Parameters - Graphical Definition

The buckling parameters may be defined or checked in a graphics environment in FS2000. This is a very efficient method for definition since a mouse may be used to define lengths and assign to elements.

This facility is available in the Design TASK.



-0-

## **5.0 Technical Specification (ISO 19902)**

This section describes in detail which of the various clauses of ISO 19902 are used by the program.

### **5.1 Design Strength**

The design strength of the element is based upon the strength specified by the appropriate material property code when the model was defined in FS2000 or when modified within the code checker.

### **5.2 Axial Tension**

Tubular sections are checked in accordance with Clause 13.2.2

The tensile stress is evaluated using the effective area ratio.

### **5.3 Axial Compression**

Tubular sections are checked in accordance with the Clause 13.2.3.1.

Column buckling strength Clause 13.2.3.2

Local buckling strength Clause 13.2.3.3

### **5.4 Shear**

Beam Shear - Sections are checked in accordance with the Clause 13.2-5.1.

Torsional shear checked in accordance with the Clause 13.2-5.2.

### **5.5 Bending**

Tubular sections are checked in accordance with the Clause 13.2.4.

### **5.6 Hydrostatic Loading**

The code checker will check the hydrostatic loading in accordance with the requirements of Clause 13.2.6.2 ([see Section 4.0](#))

The result of the check will be a unity ratio using the appropriate safety factors.

The length between stiffening rings is based on the **L<sub>lb</sub>** length parameter (see 4.1) which defaults to the element length

This unity ratio will only be shown if it is the maximum for the element.

If an element is checked individually the Critical Hoop Buckling Stress will be printed

### **5.7 Combined Loading**

Combined loading effects are checked in accordance with the Clause 13.4

Hydrostatic end cap effects are taken to be 0.5 of the hoop stress in accordance with 13.4-4.

-O-

## **5.0 Technical Specification (API RP2A LFRD)**

This section describes in detail which of the various clauses of RP2A are used by the program.

### **5.1 Design Strength**

The design strength of the element is based upon the strength specified by the appropriate material property code when the model was defined in FS2000 or when modified within the code checker.

### **5.2 Axial Tension**

Tubular sections are checked in accordance with Clause D.2.1

The tensile stress is evaluated using the effective area ratio.

### **5.3 Axial Compression**

Tubular sections are checked in accordance with the Clause D.2.2

Column buckling strength Clause D.2.2.1

Local buckling strength Clause D.2.2.2

### **5.4 Shear**

Beam Shear - Sections are checked in accordance with the Clause D.2.4.1.

Torsional shear checked in accordance with the Clause D.2.4.2.

### **5.5 Bending**

Tubular sections are checked in accordance with the Clause D.2.3.

### **5.6 Hydrostatic Loading**

The code checker will check the hydrostatic loading in accordance with the requirements of Clause D.2.5.2 ([See Section 4](#))

The result of the check will be a unity ratio using the appropriate safety factors.

The length between stiffening rings is based on the **L<sub>lb</sub>** length parameter (see 4.1) which defaults to the element length

This unity ratio will only be shown if it is the maximum for the element.

If an element is checked individually the Critical Hoop Buckling Stress will be printed

### **5.7 Combined Loading**

Combined loading effects are checked in accordance with the Clauses D.3.2 and D.3.2 for axial and bending effects.

Combined loading effects including hydrostatic effects are checked in accordance with the Clauses D.3.3 and D.3.4. Hydrostatic end cap effects are taken to be 0.5 of the hoop stress.

-0-

## APPENDIX A - Geometric Property Libraries

### Geometric Property Libraries

Element section properties may be entered directly into the Property Tables using the Geometric Properties box or may be added by selecting sections from entries from pre-defined Property Libraries.

To use the tables it is essential that the model be created in the **SI-Unit** system. The library file formats and the units of the libraries are consistent with standard structural property tables available in structural handbooks.

There are seven different property file formats for each of the following section types.

I	I beams and H columns
C	Channels
B	Box sections
T	Tee section
A	Angle sections (no bending stiffness i.e. pure strut/tie)
L	Angle section
R	Rectangular bar

The following property table files are used for standard British Rolled Sections and are supplied with the program.

Property File	Type	Description
UB.PRI	I	Universal beams
UC.PRI	I	Universal columns
RSJ.PRI	I	Joist sections
PFC.PRC	C	Channel sections (RSC also)
RHS.PRБ	B	Rectangular hollow sections
SHS.PRБ	B	Square hollow section
RST.PRT	T	Tee sections
TUB.PRT	T	Tee section cut from UBs
TUC.PRT	T	Tee sections cut from UCs
RSA.PRA	A	Angles (partially complete)
EA.PRL	L	Equal angles
UEA.PRL	L	Unequal angles

The following property table files are used for standard European sections and are supplied with the program.

Property File	Type	Description
HEA.PRI	I	I Sections
UEB.PRI	I	I Sections
HEM.PRI	I	I Sections
IPE.PRI	I	I Sections
IPN.PRI	I	I Sections

The following **USA-Unit** property table files are used for standard USA sections and are supplied with the program. The underscore (\_) identifies USA-Unit property tables.

WS_.PRI	I	W Shapes
---------	---	----------

HP_.PRI	I	HP Shapes
MS_.PRI	I	M Shapes
SS_.PRI	I	S Shapes
CS_.PRB	C	Standard Channels
MC_.PRB	C	Miscellaneous Channels
HS_.PRB	B	Rectangular tubing
WT_.PRT	T	W Tee sections
MT_.PRT	T	M Tee sections
ST_.PRT	T	S Tee sections
AA_.PRA	A	Angles (Tie/strut)
AS_.PRL	L	Angles

## Creating and Editing Property Libraries

There is no limit to the number of property library files the user may create. It is only necessary to ensure that when named, they are identified to the appropriate group by the file extension. Property files must exist in the FS2000 directory. Exceptions to this rule are model related tables (see below).

It is recommended that new property libraries be always created using the Geometric Properties Utility. This may not be possible if standard structural sections are being used and properties are require to be exact i.e. fillet radii and tapered flanges effects included.

When creating new files manually it is better to copy an existing table of a similar type and then edit it to requirements i.e. delete all existing entries but one and then add the new entries below that. The format fields of the single entry may be used as a template for the new entries.

If section properties are to be added to the files ensure that the appropriate file type is used. e.g. in the case of bearing piles ( I sections) the data could be added to either of the first three files above since the section type is similar in each case.

The filename of property files must be a 2 or 3 character name. The section designation of table entries is a numeric only designation with up to 9 characters. Within the program all property code data originating from table files are identified by the file name and the designation. e.g. UB 914419388 identifies a 914 x 419 388 kg Universal Beam.

If the designation is preceded by - ve sign, i.e. it is a negative number, the section will be treated as a welded section in the Design Code Checkers.

## Model Dependent Property Libraries

Often it will be found convenient to create custom property libraries that are related to specific models. The main advantage of this is that it enables the library to be archived with the model and eliminates the need to maintain large mixed model related libraries.

Unlike standard libraries the model related libraries must reside in the model directory and possess the model files name. The file extension is still used to identify the section type.

When model dependent table entries are used they are identified by the following ID:

MDI	I sections
MDC	Channel sections
MDB	Box sections
MDT	Tee sections
MDA	Angle sections (no bending stiffness i.e. pure strut/tie)
MDL	Angle section
MDR	Rectangular bar

Use the Geometric Properties Generation Utility to create Model Dependent Libraries.

## Geometric Library File Formats

Property libraries are plane text files (ASCII). Each section entry must be contained in one line and spaces are used to separate fields.

The unit type of the library is defined at the beginning of the first line. INCH is used to signify an USA-UNIT type library as shown below. If this is not present the library is a SI-Unit library.

```
INCH  ASTM  W Shapes
Desig D  B  t  T  r  lx  ly  Sx  Sy  J  A  H
44335 44 15.9 1.03 1.77 1.18 31100 1200 1620 236 74.7 98.5 535000
```

The file formats for each of the library types is shown below (one entry only). Only British SI-Unit libraries are shown, US-Unit library formats are identical but the units are in inches.

**UB.PRI (Type I)**

Universal Beams

```
Desig D  B  t  T  r  lx  ly  Sx  Sy  J  A  H
914419388 920.5 420.5 21.5 36.6 24.1 719000 45400 17700 3340 1730 494 88.7
```

**RSC.PRC (Type C)**

Rolled Steel Channels

```
Desig D  B  t  T  r  lx  ly  Sx  Sy  J  A  H  Cy
432102 431.8 101.6 12.2 16.8 15.2 21400 629 1210 153 61.0 83.5 .217 2.32
```

**RHS.PRB (Type B)**

Rectangular Hollow Sections

```
Desig D  B  T  lx  ly  Sx  Sy  J  A
502525 50 25 2.5 10.6 3.44 5.41 3.26 8.41 3.47
```

**SHS.PRB (Type B)**

Square Hollow Sections

```
Desig D  B  T  lx  ly  Sx  Sy  J  A
202 20 20 2 .759 .759 .951 .951 1.22 1.42
```

**RST.PRT (Type T)**

Rolled Structural Tees

```
Desig D  B  t  T  r  lx  ly  Sx  Sy  J  A  H  Cy
419457194 460.2 420.5 21.5 36.6 24.1 44100 22700 2190 1670 856 247 0 10.3
```

**RSA.PRA (Type A)**

Angles

```
DESIG D  B  T  RXX  RYY  RUU  RVV  A
50506 50 50 6 1.5 1.5 1.89 .968 5.69
```

**UEA.PRL (Type L)**

Angles

```
Desig D  B  t  T  A  Cx  Cy  lx  ly  Ruu  Rvv  Tan(Ang)
20015018 200 150 18 18 60.1 3.86 6.34 2390 1155 6.97 3.22 0.549
```

**BAR.PRR (Type R)**

RSect Type Library Entry ( mm ; cm3 ; cm4 )

```
Desig D  B  lx  ly  Sx  Sy  J  A
100025 1000.0 25.0 2.083E05 1.302E02 6.250E03 1.563E02 5.126E02 2.500E02
```

**Welded Sections**

If the section designation is negative then the section will be assumed to be of welded construction .





## **APPENDIX B - Data Files**

### **Data Files and Model Files**

Files created by the code checker are:

"modelname".ELN	Modified Parameter File (definition file)
"modelname".^"n"	Moment Coefficient File
"modelname".I"n"	Report file - Summary
"modelname".S"n"	Report file - Stresses & Allowable Stresses
"modelname".5"n"	Stress Ratio File (list of Unity Ratios in text format)

"n" is the Results Case Number

### **Modified Parameter File Format**

The .ELN file is a formatted text file that may be edited by the user. The data is defined on a line by line basis. The line is identified by the command ME. All data must be present on the line. The format for the line is shown below.

ME, *Elem*, *Lx*, *Kx*, *Ly*, *Ky*, *Llb*

<i>Elem</i>	Element Number
<i>Lx</i>	Effective Length for compressive buckling about the local xx axis.
<i>Kx</i>	Effective length for xx Comp buckling
<i>Ly</i>	Effective Length for compressive buckling about the local yy axis.
<i>Ky</i>	Effective length for yy Comp buckling
<i>Llb</i>	Unsupported Length of Compression Flange

-0-

## APPENDIX C - Batch Operation

### Command Line Operation (Batch Mode)

This appendix defines the command line options for the codechecker. For further details of command line operation refer to FS2000 Help or manual.

ISO19902 C1/C2/C3/C4/C5/C6/C7/C8/

APILRFD C1/C2/C3/C4/C5/C6/C7/C8/

- C1     The processed results case number. Ranges may be processed e.g. 1-10 would process Results Cases 1 to 10 . If cases do not exist when processing ranges a warning will be given and the process will continue to the next case.
- C2     Text file output format 2-Summary Report, 3-Short Report
- C3     Number of location on span for code check ( Setting to -3 will activate 21pt check).
- C4     Allowable stress increase factor.
- C5     Unity check ratio limit for output.
- C6     Group SET to read
- C7     Group Limit/Restriction
- C8     Groups Only switch
- C9     Subcase name
- C10    Depth of water above the mode origin/Load Factor (Hydro collapse check - [see Section 4](#))

For C6-C8 (G1 -G3) see below on using Groups for output.

Using Groups to Sort Output

- G1     Group SET to read
- G2     Group Limit/Restriction
- G3     Groups Only switch

**G1** defines the group SET to be loaded. If a SET is loaded then all node and element labels will be accompanied by their respective group attribute.

**G2** defined the Group Limit\Restriction used by the following options. If **G2** is positive then output will be restricted to only those entities with the same group number as defined the **G2** ( zero value indicates that all data will be shown). This is a restricted process option.

It **G2** is negative the Grouped output will be sorted by group up to the group limit defined by **G2**.

If **G3=1** and **G2 is negative** then entities not assigned to groups or entities assigned to Groups greater than defined by G2 will not be output. This is a restricted process option.

If **G3=0** and **G2 is negative** then entities not assigned to groups or entities assigned to Groups greater than defined by the **Group Limit/Restriction** will be output in label order following the sorted groups. All data is processed with this option.

When running in batch mode any modified element parameter file will be automatically loaded (not moment factors).

-0-



