FS2000

BS 5950 Member Design

Advanced Structural Analysis
for Windows
(c) A.E.S. Ltd 1988,2020

Email: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 Table Lists 13
4.0 Modification of Element Properties 14
4.1 Buckling Parameters 15
4.2 Moment Coefficients 16
4.3 Property Code Data 17
4.4 Buckling Parameters - Graphical Definition 19
5.0 Technical Specification 20
APPENDIX A - Geometric Property Libraries 25
APPENDIX B - Data Files and Model Files 29
APPENDIX C - Batch Operation 30

1.0 Introduction

FS 5950CodeCheck is an interactive program module that interfaces with FS2000 to provide limit state design checks in accordance with BS 5950 "Structural Use of Steelwork in Buildings" 2000 Edition.

The model in FS2000 must to be created in fundamental S.I. units (see Section 2.0).

The codechecker will check the following structural sections

- Symmetrical I sections
- Channel sections
- Rectangular hollow sections (boxes)
- Structural Angles (strut/tie only)
- Double Angles
- Structural tees
- Rectangular bar
- Sections built up by welding including plate girders will also be checked.
- Tapered sections

-o-

2.0 Creating Models in FS2000

This section describes the few requirements necessary when creating models that are going to be code checked. See Sect 4.1 regarding buckling parameters.

-o-

2.1 Units

BS5950CodeCheck reads the input files and result files of models created and analysed by FS2000. To ensure units compatibility it is essential that the model be created in fundamental S.I. units.ie

Force in N (NEWTONS)

Length in M (METRES)

Stress in N/m2

-o-

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.

Load - Partial Safety Factors

The post-processor uses load case combinations to combine multiple result cases in to a single processed result case. Load case combinations can be used to account for the loading partial safety factors used in limit state design checks.

-o-

2.3 Property Codes

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

1 Section properties of the elements (Geometric Property Code), and

2 Element steel grades (Material Property Code)

For an element to be checked, apart from CHS, the Geometric Properties MUST have been selected from an FS2000 property library . Material properties need not be selected from a library.

The section properties are identified by the codechecker by reading the section library and comparing the model Property Code designation with those in the library. When a match is found the properties are assigned from the library to the element.

If creating property libraries manually always ensure the property designations are not duplicated. The property code generator in FS2000 identifies section by the depth and weight.

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. These must also be in metres.

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

The steel grade is identified by comparing the specified yield strength of the material property codes. For the steel grades to be identified as a BS5950 Grade (Table 9) then the following values must be specified in the material property codes of FS2000.

Grade	Property Code Yield Strength N/m2
S275	275E6
S355	355E6
S460	460E6

Note : Units of stress is N/m2 in FS2000

If the steel grade is identified as either Grade S275, S355 or S460 the design strength will be evaluated in accordance with Table 9 based on the flange thickness. If not, the design strength will be based on the nominal yield strength.

If Py is greater than UTS/1.2 then UTS/1.2 will be taken as the design strength where UTS is the steel ultimate tensile strength.

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.

-o-

2.4 Structural Design Parameters

The design parameters used in a design check are obtained the basic model data defined during model creation within FS2000. Modification of theses 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 (Clause 4.8.3.3)

Llb Unsupported Length of Compression Flange

mx *Equivalent uniform moment factor*

my *Equivalent uniform moment factor*

mLT *Equivalent uniform moment factor*

mxy *Equivalent uniform moment factor*

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.

-o-

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 codechecket 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 form the code check is to plot the design utilisation ratios (UR Plot). Search or UR in the FS2000 Help index for further information on UR plots.

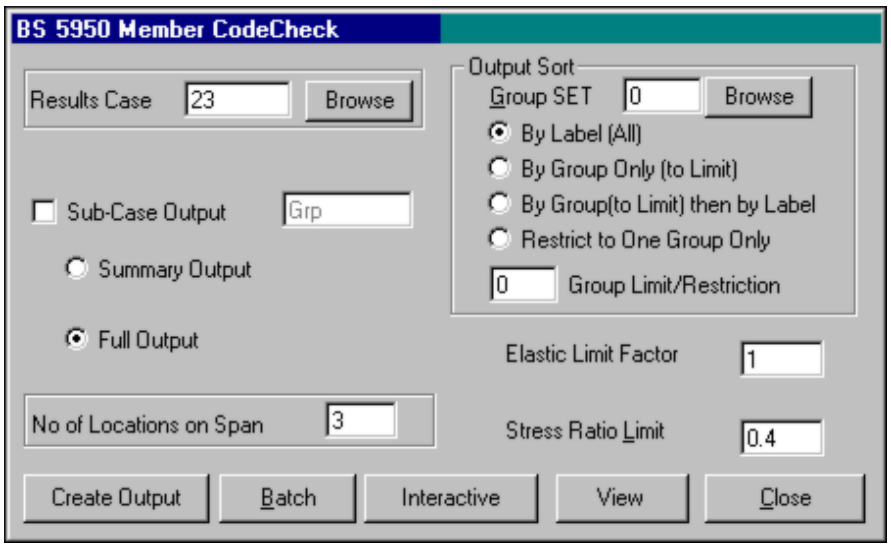
The text out from the codechecker can be view one of the following.

- 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

-o-

3.1 Operation from FS2000

When this commands 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.

Summary Output option selects the summary mode output which produces single line output (per element) which shows loading in terms of the design utilisation ratio only.

Full Output option produces the full mode output the 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. Up to 20 points along the length of the element may be specified. Generally 2 or 3 are sufficient but this is dependent on the loading on the element.

Elastic Limit Factor is used to enter the global factor described in Clause 4.2.5. The default value of 1.2 is conservative and may be increased to the limit value of 1.5.

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 **Create Output** button is used to run the cod check with the currently shown settings.

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

The **Interactive Mode** button activates Interactive operation of the CodeChecker

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

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.

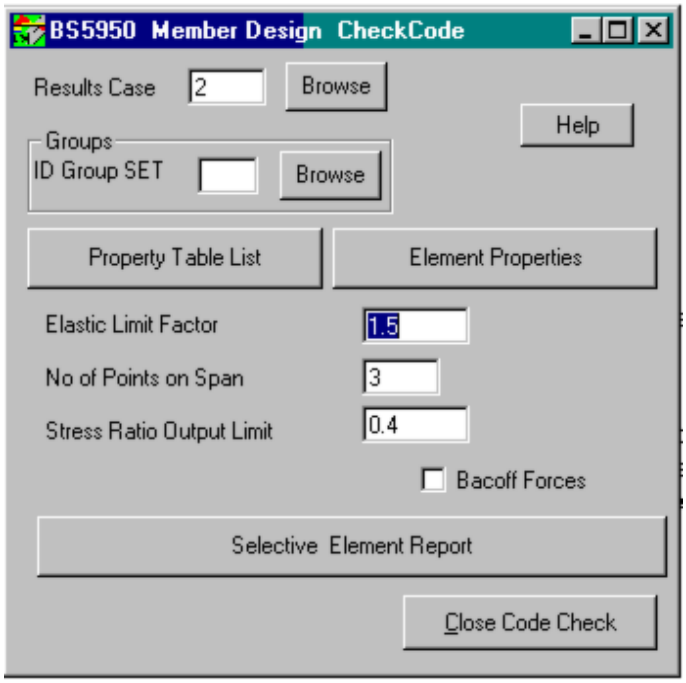
Important Note Stress ratio data created by this module will be limited to the data processed. If restricted process options is used then any Stress Ratio plots or Stress Ratio sorts which use the same results cases will be limited to the processed data. The plot or listed output will indicate if the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

-o-

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.



Apart from the **Selective Element Report** and the **Element Properties** buttons the input options of the form are identical to those of the FS2000 Member Design form described in the previous section.

The **Property Table List** will show property lists that show if the geometric amd material preperty 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

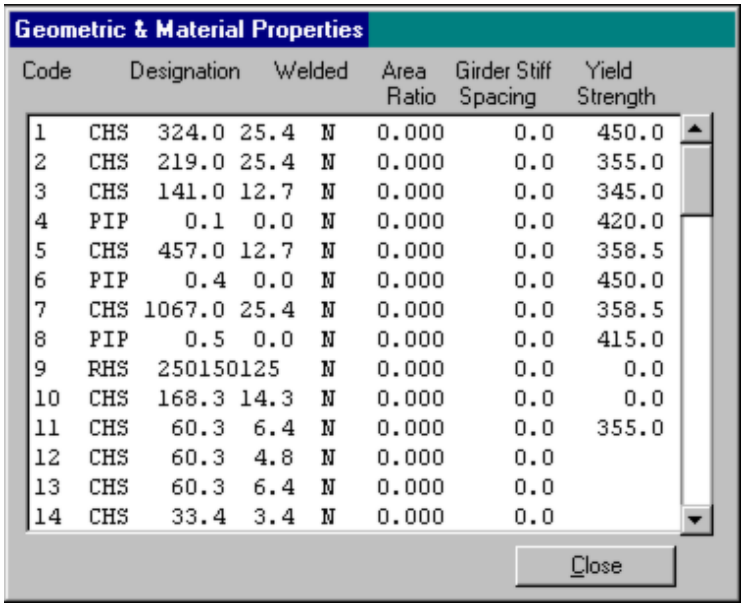
The **Element Properties** button is used to view and define elements properties. Re-defining properties enables the user to design the element for the applied loading. The re-definition of section properties is described in Section 4.0 .

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.

-o-

3.3 Property Table Lists

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



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.

-o-

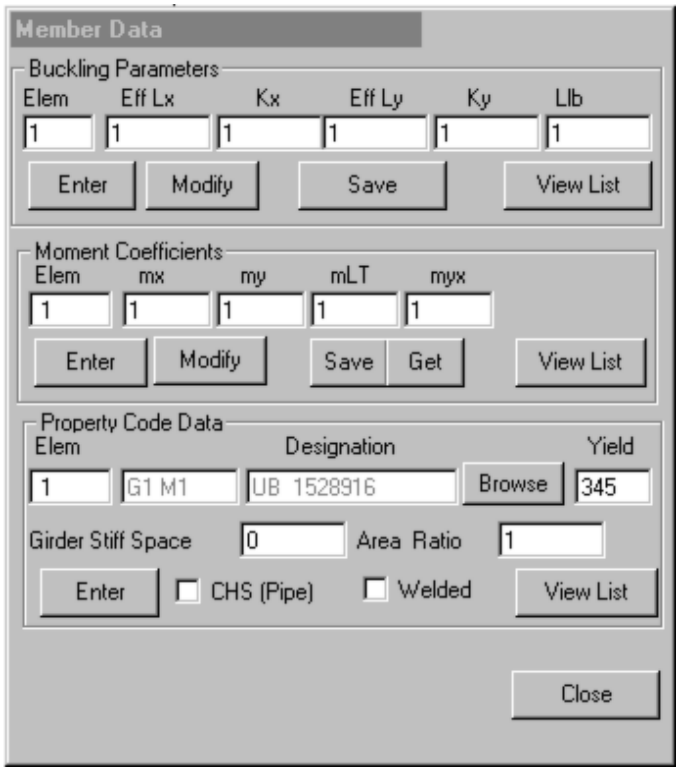
4.0 Modification of Element Properties

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



-o-

4.1 Buckling Parameters

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

Llb Unsupported Length of Compression Flange

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.

-o-

4.2 Moment Coefficients

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

mx Equivalent uniform moment factor

my Equivalent uniform moment factor

mLT Equivalent uniform moment factor

mxy Equivalent uniform moment factor

See Clause 4.8.3.3

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.

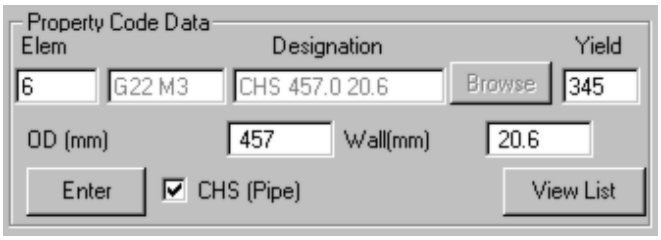
The **Get** button is used to retrieve a previously saved moment factor file. Always ensure that the retrieved case is related to the current load case file.

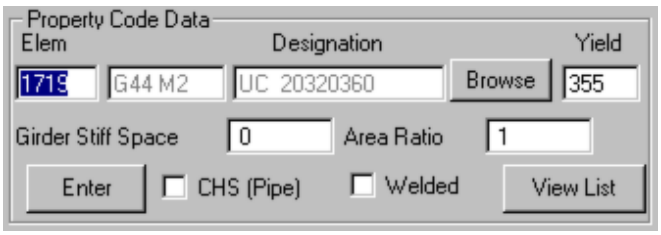
The .^'m ' files will not be loaded during command line operation. These parameters can only be used during interactive operation.

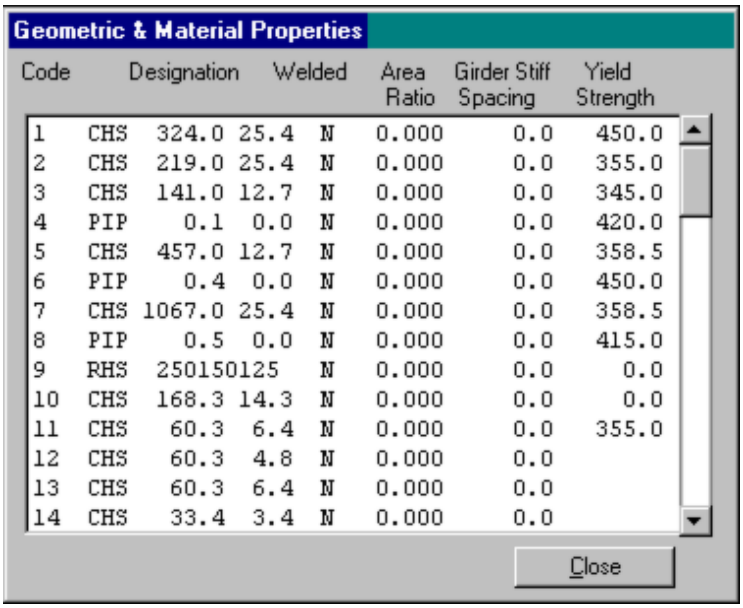
-o-

4.3 Property Code Data

This section of the Data form is used to modify parameters relating to the property codes of the elements. Note that this will change all elements with the same property codes. Changes here are temporary, they do not change the FS2000 model properties







The **ViewList** button is used to show the Geometric and Material propertied associated with the property codes of the model. Geometric property data not recognised by the codechecker 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 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 Ae/Ag ratio. Ae 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. If an A type angle is designated as welded the angle will be taken to have single bolt fixing.

The **CHS** check box is used to change the property code to that for a pipe. When checked, input boxes to 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.

-o-

5.0 Technical Specification

This section describes in detail which of the various clauses of BS 5950 are used by the program and how.

5.1.1 Design Strength

The design strength of the element is based upon the strength specified by the appropriate material property code. If the codes are recognised as an appropriate BS 5950 Grade then the design strength will be based on the flange (wall for pipes) thickness and be established in accordance with Table 9, Clause 3.1.1. Otherwise the specified strength will be used regardless of thickness.

5.1.2 Section Classification

The elements are classified in accordance with Table 11, Clause 3.5.2. The actual criteria used by the program is specified below. The limits shown in brackets are for welded sections for I, T & C sections and cold formed boxes. See section 3.5 for welded section and cold formed section definition.

	Class 1	Class 2	Class 3
	Plastic	Compact	Semi-compact
Flanges in I, T & C sections			
Bending xx & yy	b/T <= 9(8)e	b/T <= 10(9)e	b/T <= 15(13)e
Compression	b/T <= 9(8)e	b/T <= 10(9)e	b/T <= 15(13)e
Box sections			
Bending xx & yy	b/T <= 28(26)e	b/T <= 32(28)e	b/T <= 40(35)e
Compression			b/T <= 40(35)e
Webs in I, & C sections			
Bending xx	d/T <= 80e	d/T <= 100e	d/T <= 120e
Webs in boxes			
Bending yy	d/T <= 64(56)e	d/T <= 80(70)e	d/T <= 120(105)e
Webs in I, C & box sections			
Whole Compression			d/t<= 40e
Circular hollow sections			
Bending	D/t <= 40ee	D/t <= 50ee	D/t <= 140ee
Comp			D/t <= 80ee
Angles			
Bending	b/T <= 9e	b/T <= 10e	b/T <= 15e
Comp			b/T <= 15e
			b+d/T<=24e
T stems			
Bending	d/t <= 9e	d/t <= 10e	d/t <= 18e
Comp			d/t <= 18e
Rectangular bar			
Bending	d/T <= 9e	d/T <= 10e	d/T <= 15e

Dimensions d, D, b, T and t are as defined in Fig 5, Clause 3.5.1

5.1.3 Reduction of Strength/Properties in Slender Sections

Reduced strengths for slender sections are established in accordance with Clause 3.6.5. This is the alternative method to evaluating effective section properties and will be conservative for sections that exceed the Class 3 limit by other than a small margin

In cases where the section is slender due to both web and flange proportions the lower applicable strength reduction factor is used. The exception to this is for plate girders where the moment capacity is based on the higher of the flange resistance calculated from the reduced flange stress(Clause 4.4.4.2 flanges only method) and the full section resistance based on the reduced web stress (Clause 3.6.5).

5.1.4 Sections with Thin Webs

Sections where thin webs have d/t > 62 e have their strengths based on the following.

Moment capacity is based on flange resistance only (Cl 4.4.4.2).

The shear capacity of the web is based on the following (Cl 4.4.5.2)

Vcr = dtqcr

For web slenderness lw less than 0.8

qcr = .6Py

For web slenderness lw greater than 1.25

qcr = qe

Between the above the following transition is used

qcr = .6Py((13.48 - 5.6lw)/9)

where in the above

lw = (.6/Py/qe).5

qe = (.75+1/(L/d)2)(1000/(d/t)2 for L/d=<1 Appendix H

qe = (1+.75/(L/d)2)(1000/(d/t)2 for L/d>1

5.2 Shear and Bending Capacities

The shear and bending capacities of elements are evaluated in accordance with Clause 4.2 Page 21

5.2.1 Shear Capacity

	Pv = 0.6pyAv
For rolled I,H & C	Avy = tD
	Avx = tB1.8
For CHS	Av = 0.6A
For RHS & SHS	Av = (D / (D + B))A

5.2.2 Bending Capacity

The bending capacity of plastic and compact L type angles is based on

Mc = pyS .ELF

ELF is the global elastic limit factor (1.2 - 1.5 in Clause 4.2.5).

Bending Capacity with Low Shear

In accordance with Clause 4.2.5.2 For Fv <= 0.6Pv

Plastic & compact sections Mc = pyS

Semi-compact sections Mc = pyZ

Slender sections Mc = pyZ

Bending Capacity with High Shear

In accordance with Clause 4.2.5.3 For Fv> 0.6Pv

Plastic & compact sections Mc = py(S - .Sv)

Semi-compact sections Mc = py(Z - .Sv / 1.5)

Slender sections Mc = pyr((Z - .Sv / 1.5)

where = [2.(Fv/ Pv) - 1]2

Note : py and pyr are the design and reduced design strength

5.2.3 Torsional Shear

Although not a code requirement, the program checks for torsional loading in tubular and box sections. The torsional capacities are based on

Limiting Torsional Moment = 0.6pyZt

For Tubes Zt = 2Z For boxes Zt=2Tw(B-Tw)(D-Tf)

5.3 Torsional Lateral Buckling Capacities of Beams

The bending capacities of elements about their major axis in terms of torsional lateral buckling is evaluated in accordance with Clause 4.3.

The structural model elements may not fully represent the loading and restraint configuration of the actual beam. In BS5950 the mLT parameter is used for this purpose. Therefore the user is required to input the appropriate value depending upon the loading and restraint condition of the actual beam. The program default value is unity. This values is conservative value.

Similarly the effective length between lateral restraints is also required to be defined by the user. The default value is the element length. This may not be conservative and therefore should always be checked to ensure this reflects the actual restraint condition of the structure.

Mx <= Mb / mLT

The above two methods use the following equations:

Plastic & compact sections Mb = pb Sx

Semi-compact sections Mb = pb Zx

Slender sections Mb = pb Zx

For slender section the pb is based on the reduced strength.

The evaluation of pb is done using the equations given in Annex B.2. The Equivalent Slenderness for the different section types are evaluated in accordance with Annex B.2. These equations have not been repeated here.

The slenderness factor v for RST and angle sections is conservatively based on a negative monsymmetric index.

The buckling resistance of angles are checked using moments related to the principle axis. The moments shown in the output relate to the principle axis.

The buckling resistance of rectangular bar is based on the x-x axis being the major axis.

5.4 Tensile Load Capacities

The axial capacities of members under tensile loading is in accordance with Clause 4.6.0.

Pt = Aepy

The user defines the effective area in terms of area ratio between effective area and gross area ie Ae/A.

The user defines this ratio in Option 4 of the main menu. The program default value is unity.

5.5 Compressive Load Capacities

The axial capacities of members under tensile loading is in accordance with Clause 4.7

Pc = Agpc

The compressive design strength is a function of the slenderness ratio and the type of strut ie UC etc. The strut curves are based on Table 23. All hollow sections are taken to be hot finished.

The compressive strength is evaluated about both major axes of the element ie px and py.

The slenderness ratio is dependent upon the effective length of the strut. The program uses the length of the element as the default values. The user may redefine the effective lengths to reflect the true restraint condition of the strut. It is essential that the appropriate length is used as the default values may be non-conservative.

In the case of A type angles the slenderness is established in accordance with Clause 4.7.10.2. Lx and Ly are used to define the buckling length for the geometric axis. LIb is used to define the buckling length for the minor principle axis. If Kx is specified as 0.85 (default) then the long leg if the angle is fixed. If Ky is specified as 0.85 then the short leg if the angle is fixed. Note that in the program Kv is the product of Kx and Ky therefore if one is 0.85 the other must be 1.0. If the designation is specified as welded (see Sect 4.3), the angle will assumed to have single bolt fixing (condition C). This is the default condition.

In the case of L angles the capacities are shown in terms of their principle axis. x-x is the u-u major principle axis and y-y the vv minor principle axis.

The compressive design strength is evaluated using the equations defined in Annex C.

5.6 Combined Load Capacities

The load capacities of elements subject to combined loading are established in accordance with Clause 4.8 .

5.6.1 Reduced Moment Capacity

When a section is loaded under both axial and bending loads there may be a significant shift of the neutral axis depending upon the level of loading. To take this effect into account a reduced plastic modulus Sr may be used since the code provides an alternative interaction formula which uses this reduced moduli. This alternative will give greater economy but is only applicable to plastic or compact sections.

Combined loading on angles, tees and channels use only the simplified axial/bending interaction formula and take no account of reduced moment capacity.

For I sections Sr is established using the equations given in Annex J2.

For Circular hollow sections

Sr = S-S[1-cos(nPi/2)] for cos(nPi/2)>=[1-(2r/R=r))2].5

For box sections

Srx=Sx - A2n2/8t for n <= (D-2t)2t/A

Sry=Sy - A2n2/8t for n <= (B-2t)2t/A

where in the above n = F/Agpy

5.6.2 Tension Members with Moments

The following interaction ratios are evaluated for local capacity check. Clause 4.8.2 .

The semi-compact and slender sections (all sections)

F/Aepy + Mx/Mcx + My/Mcy <= 1

For plastic or compact sections (economic alternative)

(Mx/Mrx)z1 + (My/Mry)z2 <= 1

Z1 = 2 Z2 = 1 for I sections

Z1 = Z2 = 5/3 for Box sections and rectangular bar

Z1 = Z2 = 1 for all other sections

Z1 = Z2 = 2 for circular hollow and solid sections

5.6.3 Compression Members with Moments

The following interaction ratios are evaluated for local capacity check. Clause 4.8.3.

For semi-compact and slender sections (all sections)

F/Agpy + Mx/Mcx + My/Mcy <= 1

For plastic or compact sections (economic alternative)

(Mx/Mrx)z1 + (My/Mry)z2 <= 1

The following interaction ratios are evaluated for overall buckling capacity check.

Simplified approach Clause 4.8.3.3.1

Fc/Pc + Mx/pxZx+ My/pyZy <= 1

Fc/Pcy +mxMx/Mb + myMy/pyZy <= 1

For I, CHS and RHS the more exact method of 8.3.3.2 and 8.3.3.3 are employed.

In cases where only one loading component is present (plastic/compact sections only) the resultant utilisation from the simplified approach will dominate . To prevent this from occurring simplified combined checks on compact sections are only evaluated for component utilisation greater than 5%. This anomaly does not occur with the more exact method.

-o-

APPENDIX A - Geometric Property Libraries

Geometric Property Libraries

Units – Two types of section libraries are used, SI-Unit libraries and USA-Unit libraries. The library file formats and the units of the libraries are consistent with standard structural property tables available in structural handbooks.

To use the tables it is essential that the model be created in either the **SI-Unit** system or the **USA-Unit** system. A library unit conversion is used by the program to enable access to both types of section library, regardless of the model unit system.

There are nine different property file formats for each of the following section types. The last character of the file extension of the library file identifies the 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
D	Double Angle
1	Type 1 Beam section (unsym I beam)
2	Type 2 Beam section (unsym I beam)

The following **SI-Unit** 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.PRB	B	Rectangular hollow sections
SHS.PRB	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
DEA.PRD	D	Double equal angle
DUA.PRD	D	Double unequal angle

The following property **SI-Unit** 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
DE_.PRD	D	Double Angles
DU_.PRD	D	Double 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
MDD	Double Angle section
MDD	Double 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 data 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     Ix     Iy     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 shows, US-Unit library formats are identical but the units are in inches.

Angle Principle Axis

The Ix and Iy for Type L angles section can be either geometric or principle axis I values. The program will set the stress points value accordingly. Note that this does affect the design codes checks which always use principle axis regardless of the I value used in the library.

UB.PRI (Type I)

Universal Beams

Desig D B t T r Ix Iy 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 Ix Iy 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 Ix Iy 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 Ix Iy 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 Ix Iy 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 (No bending stiffness)

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 Ix Iy Ruu Rvv Tan(Ang)

20015018 200 150 18 18 60.1 3.86 6.34 2390 1155 6.97 3.22 0.549

DEA.PRD (Type D)

Double Angles

Desig	D	B	t	T	s	Ix	Iy	Sx	Sy	J	A	H	Cy
10010012	100.0	100.0	12.0	12.0	10.0	414	939.9	0	0.	21.66	45.4	0.0	2.90

BAR.PRR (Type R)

RSect Type Library Entry (mm ; cm3 ; cm4)

Desig D B Ix Iy Sx Sy J A

100025 1000.0 25.0 2.083E05 1.302E02 6.250E03 1.563E02 5.126E02 2.500E02

BMS.PR1 (Type 1 Beam) or BMS.PR2 (Type 2 Beam)

1 Section Library Entry (mm ; cm3 ; cm4)

Desig D B Bb t T Tb Ix Iy Sx Sy J A H Cx Cy

20032 200 200 100 6 10 10.0 2780 750.3 291.9 126.6 11.30 40.80 0 10 7.67

-o-

APPENDIX B - Data Files and Model Files

Data Files and Model Files

"modelname".ELN	Modified Parameter File
"modelname".^"n"	m & n Moment Coefficient File
"modelname".I"n"	Report file - Summary
"modelname".S"n"	Report file - Loads & allowables
"modelname".M"n"	Unity Ratio File

"n" is the Results Case number

Modified Parameter File Format

The .ELN file is a formated text file. Although not recommended it may be edited by the user. The data is defined on a line by line basis. The top header lines are for information only.

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*

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

-o-

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.

CODE C1/C2/C3/C4/C5/C6/C7/C8/C9/

- 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 & short report output
- C4 Elastic limit factor (global) Clause 4.2.5.1
- C5 Unity check ratio limit for output.
- C6 Group SET to read
- C7 Group Limit/Restriction
- C8 Groups Only switch
- C9 Subcase name

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 folowing 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.

-o-