FS2000

EC3 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

FS Eurocode3 (FS-EC3) is an interactive program module that interfaces with FS2000 to provide limit state design checks in accordance with DD ENV 1993-1-1:2005 Design of Steel Structures Part 1.1 General Rules and Rule for Buildings.

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

The program will undertake design checks on structural models

The EC3 codechecker will check the following structural sections

I sections Doubly Symmetrical

I sections Symmetrical about minor axis

Channel sections

Rectangular hollow sections (boxes)

Circular hollow sections (& Round bar)

Structural angles (strut/tie only)

Structural angles (with bending)

Double Angles

Structural tees

Rectangular bar

Sections built up by welding including plate girders will also be checked.

-o-

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.

-o-

2.1 Units

FS-EC3 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 Results & Partial Safety Factors

Before the design codechecker can be run the raw analysis case results must be processed in the Post-Processor of FS2000. The Post-Processor converts raw result cases into processed result cases.

Resistance - Partial Safety Factors

The default resistance factors used by the program are:

Gamma0 = 1.1

Gamma1 = 1.1

Gamma2 = 1.25

These default values will be copied to a model dependent file MODEL.UEC3RF. These parameters then become model dependent and can be changed by editing the MODEL.UEC3RF file.

The default values may be redefined by creating a EC3RF.DAT file in the FS2000 folder. This is a simple text file with the data in sequential order as shown below. The values below are those currently recommended when using EC3 for building design in the UK.

1.0

1.0

1.25

TI

Combined Load Check

TI is a global switch for the combined loading interaction (see Sect 5.6.1) check. This value should be set to 0 or 1. 0 is the default and will be used if the parameter is not present in the file.

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 the limit state design checks.

-o-

2.3 Property Codes

The main model parameters which 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.

For the steel grades to be identified as an EN 10025-2 EC3 Grade then the following values for material yield must be specified in the material property codes of FS2000.

Grade	Property Code Yield Strength N/m2
S235	235E6
S275	275E6
S355	355E6
S450	440E6

Note : Units of stress is N/m2 in FS2000

If the steel grade is identified as either S235, S360, S430 or S510 the design strength will be evaluated in accordance with Clause 3.2.1 Table 3.1, based on the flange thickness. The ultimate strength will also be taken as that defined in Table 3.1

If the steel grade is not identified, the design strength will be based on the yield strength and ultimate obtained from the material property code.

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 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

$\propto_{xx}$ Imperfection factor xx axis

$\propto_{yy}$ Imperfection factor xx axis

Lateral-Torsional Buckling

Llb Unsupported Length of Compression Flange

Cmx *Moment factor Annex BB*

Cmy *Moment factor Annex BB*

CmLT *Moment factor Annex BB*

(k/C).5** *Moment distibution/Effective length factor*

kc *Moment distibution correction factor*

$\propto$ *Imperfection 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.

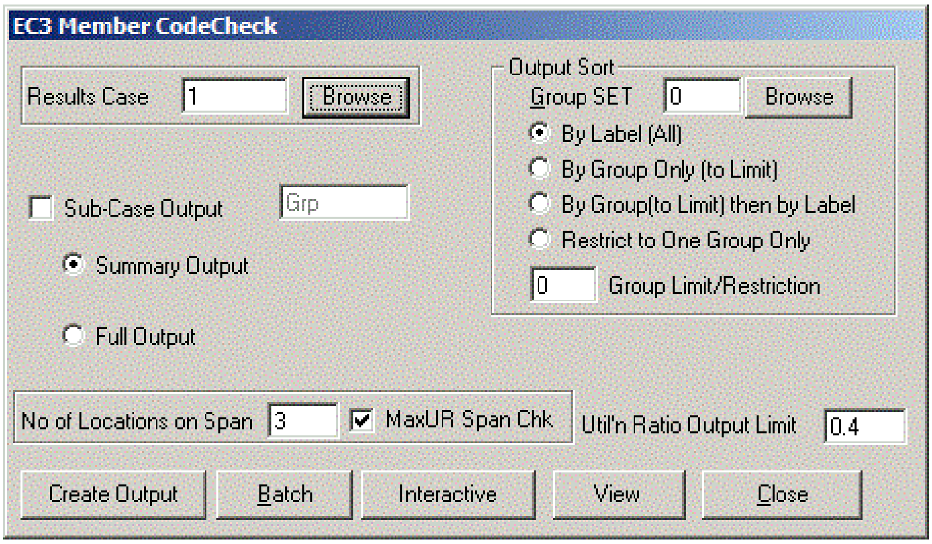
The text out from the codechecker can be view one of the following.

- 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 e.g. 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 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.

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 **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 **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.

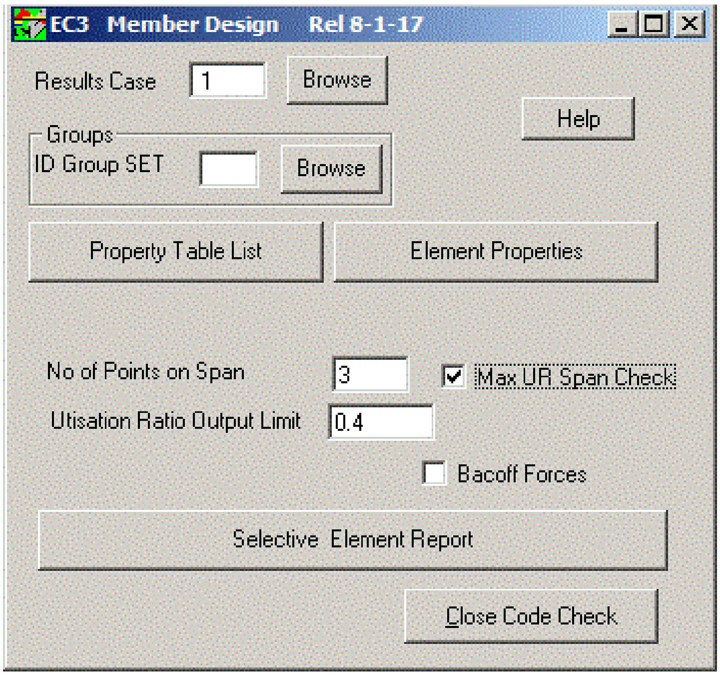
Important Note: Stress ratio data created by this module will be limited to the data processed. If restricted process options is used then any Stress Ratio plots or Stress Ratio sorts that use the same result case will be limited to the processed data. The plot or listed output will indicate if the output 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 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.

-o-

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	

Close

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-

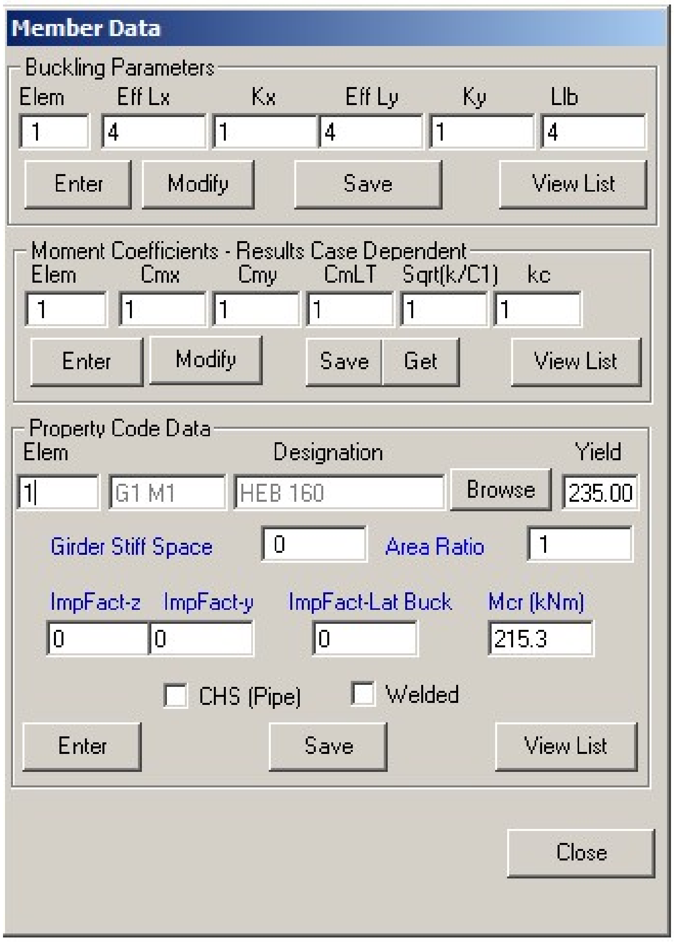
4.0 Modification of Element Properties

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

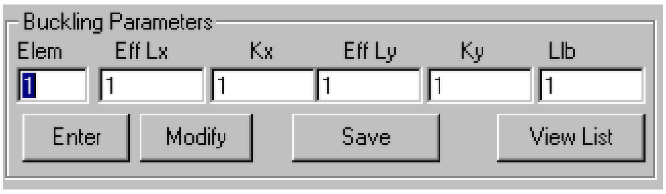
The Property Code data section relates to data defined by Geometric Property Code and Material Property Code reference and is described in Section 4.3



-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 - Lateral-torsional buckling, see Section 4.2

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

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.

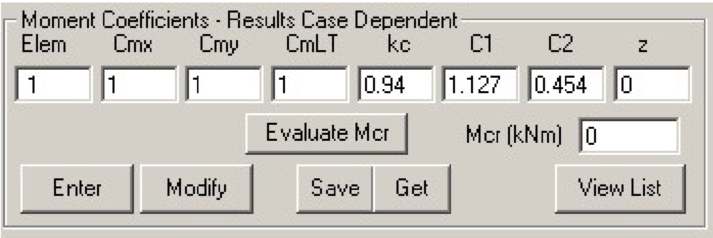
The **Save** button is used to save all current element buckling data entered to the definition file (.ELN file). This is a formatted text file which should be included in any analysis report. If one exists it will always be loaded when the code check module is started. The file may be edited. The instruction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



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 .^E'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.

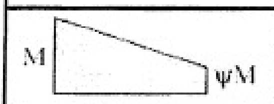
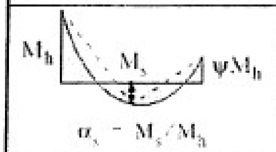
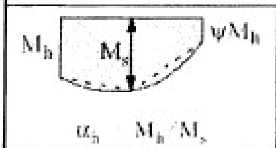
Combined column buckling checks Clause 6.3.3

Cmx Annex BB

Cmy Annex BB

CmLT Annex BB

Table B.3: Equivalent uniform moment factors C_m in Tables B.1 and B.2

Moment diagram	range		C_{my} and C_{mz} and C_{mLT} uniform loading	concentrated load
 M, ψM	$-1 \le \psi \le 1$		$0,6 + 0,4\psi \ge 0,4$	
 M_h, M_s, ψM_h; $\alpha_s = M_s/M_h$	$0 \le \alpha_s \le 1$	$-1 \le \psi \le 1$	$0,2 + 0,8\alpha_s \ge 0,4$	$0,2 + 0,8\alpha_s \ge 0,4$
	$-1 \le \alpha_s < 0$	$0 \le \psi \le 1$	$0,1 - 0,8\alpha_s \ge 0.4$	$-0,8\alpha_s \ge 0,4$
		$-1 \le \psi < 0$	$0,1(1-\psi) - 0,8\alpha_s \ge 0,4$	$0,2(-\psi) - 0,8\alpha_s \ge 0,4$
 M_h, M_s, ψM_h; $\alpha_h = M_h/M_s$	$0 \le \alpha_h \le 1$	$-1 \le \psi \le 1$	$0,95 + 0,05\alpha_h$	$0,90 + 0,10\alpha_h$
	$-1 \le \alpha_h < 0$	$0 \le \psi \le 1$	$0,95 + 0,05\alpha_h$	$0,90 + 0,10\alpha_h$
		$-1 \le \psi < 0$	$0,95 + 0,05\alpha_h(1+2\psi)$	$0,90 - 0,10\alpha_h(1+2\psi)$

For members with sway buckling mode the equivalent uniform moment factor should be taken $C_{my} = 0,9$ or $C_{Mz} = 0,9$ respectively.

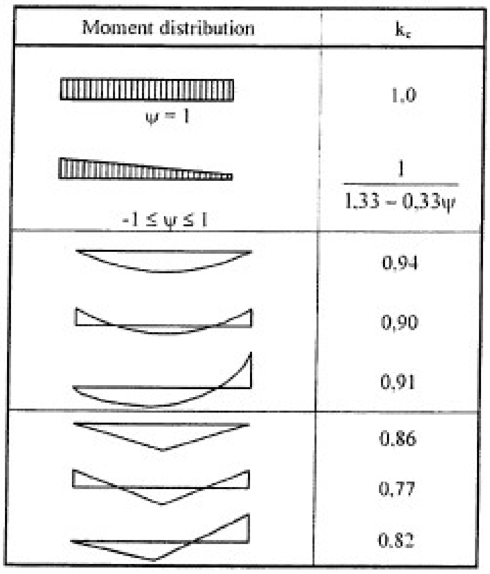
C_{my}, C_{mz} and C_{mLT} should be obtained according to the bending moment diagram between the relevant braced points as follows:

moment factor	bending axis	points braced in direction
C_{my}	y-y	z-z
C_{mz}	z-z	y-y
C_{mLT}	y-y	y-y

Lateral-torsional buckling checks Clause 6.3.2

kc accounts for the moment distribution between lateral restraints and is obtained from Table 6.6 in EC3 and is used to evaluate f in Clause 6.3.2.3.

Table 6.6: Correction factors k_c



Moment distribution	k_c
$\psi = 1$	1,0
$-1 \le \psi \le 1$	$\frac{1}{1,33 - 0,33\psi}$
(diagram)	0,94
(diagram)	0,90
(diagram)	0,91
(diagram)	0,86
(diagram)	0,77
(diagram)	0,82

C1, C2 and z are use to evaluate the Elastic critical moment Mcr using the formula given Section 5.3. Typical values for C1 and C2 are given below.

z is the distance from the point of load application to the shear centre. +ve destabilises and -ve stabilises.

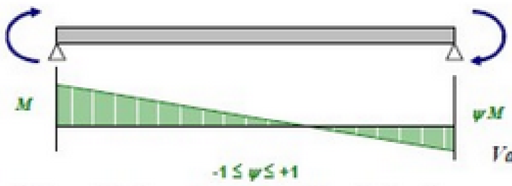
C1 is used when using the Equivalent slenderness approach.

If C2 is specified as -1, Mcr for doubly symmetric sections will be evaluated using the Equivalent slenderness approach

If the Mcr value is specified as a non zero value then this will be used in preference to the above two methods.

See Section 5.3 for more information.

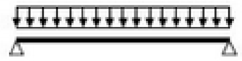
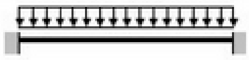
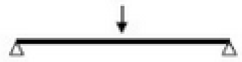
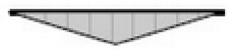
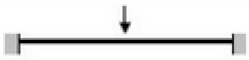
Moment Coefficents (Ref:SN003a-EN-EU)



Values of C_1 for end moment loading (for $k = 1$)

ψ	C_1
+1,00	1,00
+0,75	1,14
+0,50	1,31
+0,25	1,52
0,00	1,77
-0,25	2,05
-0,50	2,33
-0,75	2,57
-1,00	2,55

Values of factors C_1 and C_2 for cases with transverse loading (for $k = 1$)

Loading and support conditions	Bending moment diagram	C_1	C_2
		1,127	0,454
		2,578	1,554
		1,348	0,630
		1,683	1,645

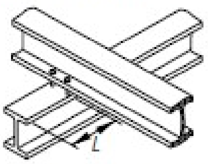
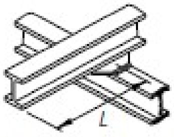
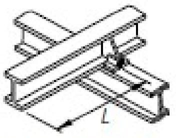
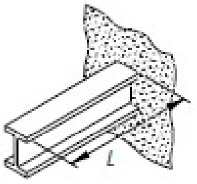
Effective Lengths for Lateral Buckling

The following extract from BS5905 give recommendations regarding the effective lengths to be used for lateral buckling checks.

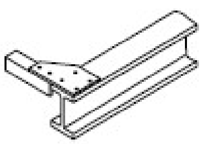
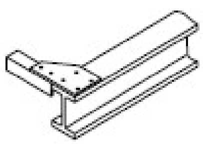
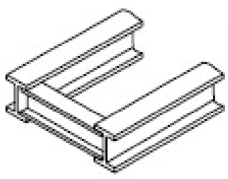
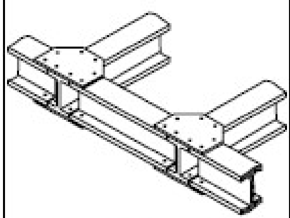
Table 13 — Effective length L_E for beams without intermediate restraint

Conditions of restraint at supports		Loading condition	
		Normal	Destabilizing
Compression flange laterally restrained. Nominal torsional restraint against rotation about longitudinal axis, as given in 4.2.2.	Both flanges fully restrained against rotation on plan.	$0.7L_{LT}$	$0.85L_{LT}$
	Compression flange fully restrained against rotation on plan.	$0.75L_{LT}$	$0.9L_{LT}$
	Both flanges partially restrained against rotation on plan.	$0.8L_{LT}$	$0.95L_{LT}$
	Compression flange partially restrained against rotation on plan.	$0.85L_{LT}$	$1.0L_{LT}$
	Both flanges free to rotate on plan.	$1.0L_{LT}$	$1.2L_{LT}$
Compression flange laterally unrestrained. Both flanges free to rotate on plan.	Partial torsional restraint against rotation about longitudinal axis provided by connection of bottom flange to supports.	$1.0L_{LT} + 2D$	$1.2L_{LT} + 2D$
	Partial torsional restraint against rotation about longitudinal axis provided only by pressure of bottom flange onto supports.	$1.2L_{LT} + 2D$	$1.4L_{LT} + 2D$
D is the overall depth of the beam.			

Table 14 — Effective length L_E for cantilevers without intermediate restraint

Restraint conditions		Loading conditions	
At support	**At tip**	**Normal**	**Destabilizing**
a) Continuous, with lateral restraint to top flange	1) Free	$3.0L$	$7.5L$
	2) Lateral restraint to top flange	$2.7L$	$7.5L$
	3) Torsional restraint	$2.4L$	$4.5L$
	4) Lateral and torsional restraint	$2.1L$	$3.6L$
b) Continuous, with partial torsional restraint	1) Free	$2.0L$	$5.0L$
	2) Lateral restraint to top flange	$1.8L$	$5.0L$
	3) Torsional restraint	$1.6L$	$3.0L$
	4) Lateral and torsional restraint	$1.4L$	$2.4L$
c) Continuous, with lateral and torsional restraint	1) Free	$1.0L$	$2.5L$
	2) Lateral restraint to top flange	$0.9L$	$2.5L$
	3) Torsional restraint	$0.8L$	$1.5L$
	4) Lateral and torsional restraint	$0.7L$	$1.2L$
d) Restrained laterally, torsionally and against rotation on plan	1) Free	$0.8L$	$1.4L$
	2) Lateral restraint to top flange	$0.7L$	$1.4L$
	3) Torsional restraint	$0.6L$	$0.6L$
	4) Lateral and torsional restraint	$0.5L$	$0.5L$

Tip restraint conditions

1) Free	2) Lateral restraint to top flange	3) Torsional restraint	4) Lateral and torsional restraint
			
(not braced on plan)	(braced on plan in at least one bay)	(not braced on plan)	(braced on plan in at least one bay)

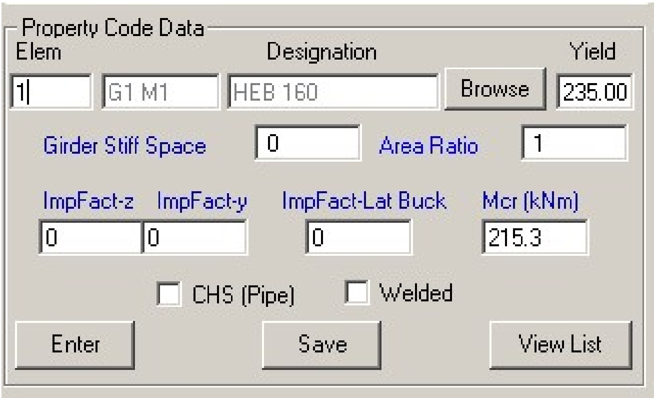
-o-

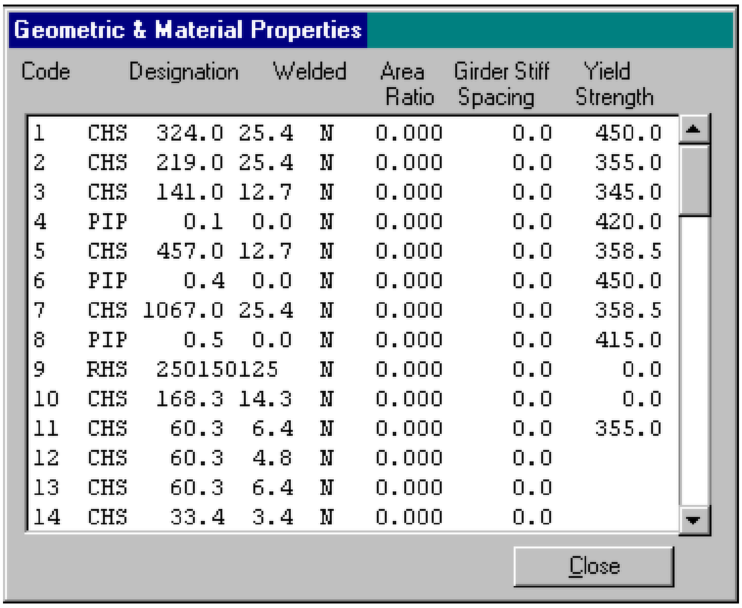
4.3 Property Code Data

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 geometric property code.

The **Enter** button will only change the Geometric and Material Properties in the the design checker, no changes will be be made the the model definition.

The **Save** button can be used to save the code checker specific properties. In this code only the variables shown in blue are saved. This design data is save to the Element Geometric Design Properties file (<model>.EDP).





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 **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 Geometric and Material 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 **Girder Stiff Space** is the distance between plate girder stiffeners

The **ImpFact-x** box is the column buckling curve imperfection factor x-x axis . A zero value will use the program selected value from Table 6.1.

The **ImpFact-y** box is the column buckling curve imperfection factor y-y axis. A zero value will use the program selected value from Table 6.1.

The **ImpFact-Lat** box is the lateral buckling curve imperfection factor. A zero value will use the program selected value from Table 6.5. If this value is defined as a negative value for I or H sections then Eqn 6.56 will be used to evaluate the lateral buckling reduction factor using the ABS value of this factor. A positive value would use Eqn 6.57.

The **Mcr kNm** box is used to define the critical elastic moment Mcr.

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 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.

The **Save** button is used to save the element property data (by geometric property code reference), and only those whose parameters have been modified. Only data with the blue labels is saved. The parameters are saved to a <modelname>.EDP file. This data will be included in the formatted modal data.

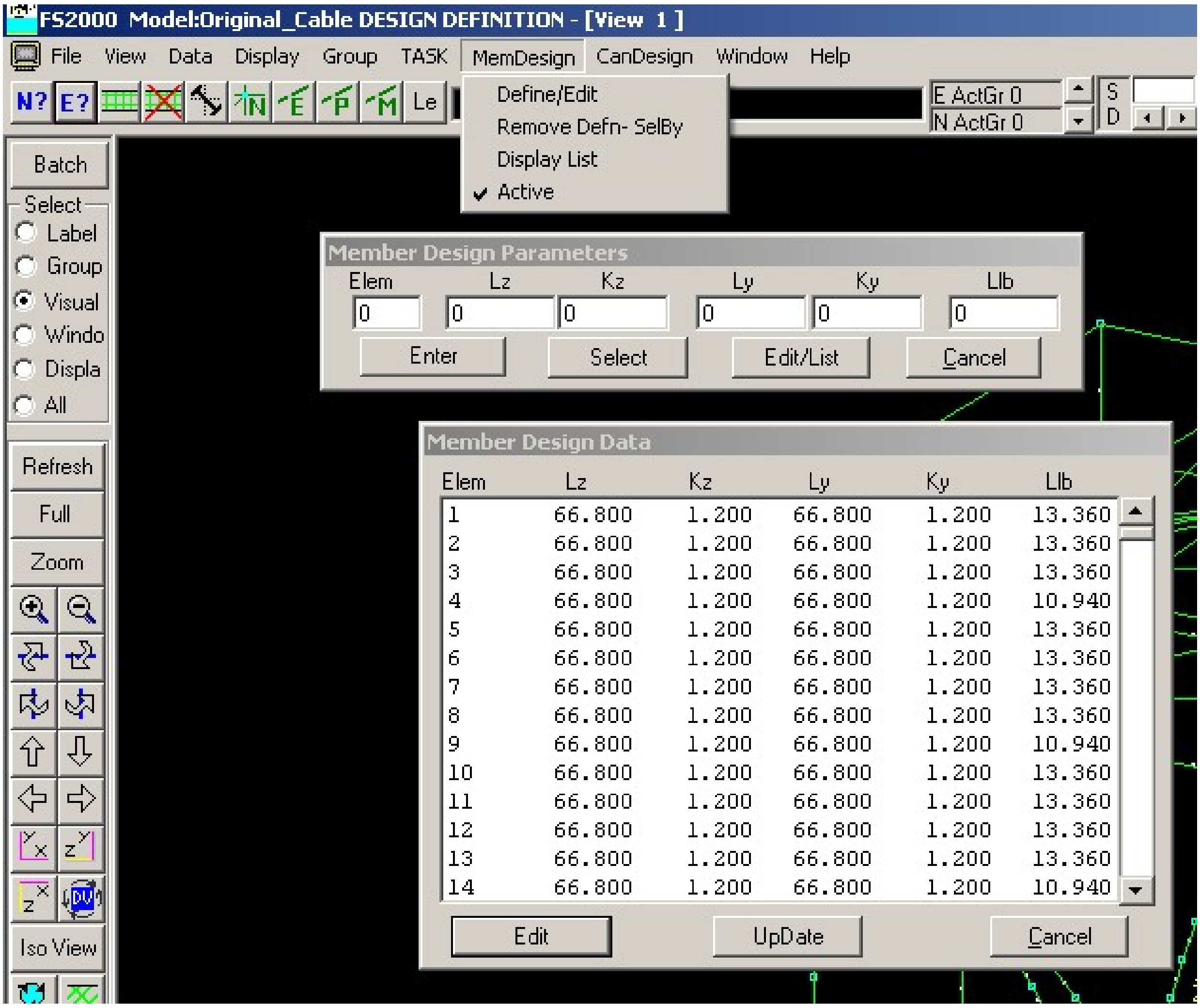
-o-

4.4 Buckling Parameters - Graphical Definition

The element buckling parameters may be defined or checked in a graphics environment in FS2000. This is the recommend method for data definition.

This is a very efficient method for definition since a mouse may be used to define lengths (node query) and then be assigned 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 EC3 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 EC3 Grade then the design strength will be based on the flange (wall for pipes) thickness and be established in accordance with Clause 3.2.1 Table 3.1. Otherwise the specified strengths will be used regardless of thickness. See 2.3

5.1.2 Section Classification

The elements are classified in accordance with Cl 5.5, Table 5.2. The specific limits used by the program are specified below. . Section not complying with Class 3 limits are classified as Class 4 (slender)

	Class 1	Class 2	Class 3
	Plastic	Compact	Semi-compact
Flanges in I, T & C sections			
Bending xx & yy	c/t <= 9e	c/t <= 10e	c/t <= 14e
Compression	c/t <= 9e	c/t <= 10e	c/t <= 14e
Box sections			
Bending xx & yy	b/t <= 33e	b/t <= 38e	b/t <= 42e
Compression	b/t <= 42e	b/t <= 42e	b/t <= 42e
For RHS sections b = b-3t in the above.			
Webs in I, T, C & box sections			
Bending xx	d/t <= 72e	d/t <= 83e	d/t <= 124e
Webs in boxes			
Bending yy	d/t <= 72e	d/t <= 83e	d/t <= 124e
Webs in I, C & box sections			
Whole Compression	d/t <= 33e	d/t <= 38e	d/t<= 42e
Circular hollow sections			
Bending & Comp	d/t <= 50e.e	d/t <= 70e.e	d/t <= 90e.e
Angles			
Bending & Comp			h/t <= 15e
			b+h/2t<=11.5e
T stems			
Bending & Comp	d/t <= 9e	d/t <= 10e	d/t <= 14e
Rectangular bar			
Bending & Comp	d/2t <= 9e	d/2t <= 10e	d/2t <= 14e

5.1.3 Reduction of Strength in Class 4 Slender Sections

In the program reduction factors are used to directly factor the design strength assuming that reduction factor is applicable to all elements of the section. This differs from the code approach in which the capacities are based on effective section properties.

Note: Unless the Class 3 section limit is exceeded by only a small margin, the use of the reduction factor approach can be rather conservative.

The reduction factor is based on the limiting b/t (d/t) ratio for a Class 3 cross section.

Reduction factor = (Class 3 Limit Ratio / Actual Ratio)**2

In cases where the section is slender due to both web and flange proportions the lower applicable strength reduction factor is used.

In cases where the web is slender the section moment and axial resistance is based on the greatest of the resistance obtained considering flange only action or the full section resistance using the reduction factor.

5.2 Shear and Bending Capacities

The shear and bending capacities of elements are evaluated in accordance with Clauses 5.4.6 and 5.4.7.

5.2.1 Shear Capacity

Vp = 0.5773.fyAv./ mo

For rolled I,H & C	Avy = 2.b.tf + (tw + 2r).tf	not less than hw.tw
	Avx = 2.tf.b	
For rolled T	Avy = 0.9(A-b.tf)	
For welded I,H & C	Avy = tf.d	
	Avx = 2.tf.b	
For CHS	Av = 0.6366.A	
For RHS & SHS	Av = Ab / (b + h)	
For welded boxes	Avy = 2.tf.d	
	Avx = 2.tw.b	

Thin Webs - Shear Buckling

If the d/t > 72e then the buckling shear capacity of the web is based on the following (Cl. 5.6.3 Simple post-critical method). The distance between transverse stiffeners can be defined only when using the program in interactive mode. These changes can be saved. The default is no intermediate transverse stiffeners.

V = d.t.q / mo

For web slenderness ls less than 0.8

q = fy / ls

For web slenderness ls greater than 1.2

q = (0.9 / ls).(fy / ls)

Between the above the following transition is used

q = (1 - 0.625(ls - 0.8).(fy / ls)

where in the above

ls =(d/t)/(37.4.e.SGRT(kt))

No transverse stiffeners kt = 5.34

Stiffeners at a/d < 1 kt = 4 + 5.43/ (a/d)**2

Stiffeners at a/d >= 1 kt = 5.34 + 4/ (a/d)**2

5.2.2 Bending Capacity with Low Shear

In accordance with Clause 6.2.8(2) For Vs <= 0.5Vp

Class 1 & 2 Sections	Mc = fy.Wpl /mo
Class 3 sections	Mc = fy.Wel / mo
Class 4 sections	Mc = fyWeff / m1

In the program Weff is replaced by Wel.SlendRedFact (see 5.2.3)

Note then L type angles have the moment transformed to the principle axis and all output refers to these axis.

5.2.3 Bending Capacity with High Shear

In accordance with Clause 6.2.8(3) For Vs > 0.5Vp

For I,H & C and boxes the following is used

Mv = Mf + (Mpl - Mf)(1 - R)

where R = (2.Vs/Vpl - 1)**2

Mf = Moment resistance with shear area removed

Mpl = Moment resistance in the absence of shear

In boxes the bending resistance orthogonal to the shear load is evaluated using a reduced strength using the lower of the resistances evaluated from the above equation and the following equation.

For other sections ** Mv = fy.Wpl.(1-R) /mo but not more than Mc (6.2.8(1))

** Note that the strength reduction is conservatively applied to the whole section.

In all cases the bending capacity with high shear cannot be higher than that with low shear.

High Shear in Thin Webs

For thin webs d/d>69 the section moment capacity is based on the section's flanges only therefore no reduction is web bending is required.

For CHS the shear area for the reduced strength is taken to be adjacent to the neutral axis. For CHS subjected to bi-axial loading the reduced shear strength is applied to both axis of bending which results in bending direction orthogonal to the shear force being reduced the most. If the resultant shear load is not within 10% of a principle axis this lower value will be applied to both bending axis. The shear load used in the reduction evaluation is always the resultant shear.

Note then L type angles have the moment transformed to the principle axis and all output refers to these axis.

5.2.4 Torsional Shear

The program checks for torsional loading in tubular, box sections and open sections. For open sections the Zt is taken from the model geometric property code Zt value (FS2000 default as below).

The torsional capacities are based on:

Limiting Torsional Moment = 0.5773.fy.Zt./ mo

For Tubes Zt = 2Z

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

For open sections (St Venant torsion) Zt = J/t where t is the larger thickness

J = Sum of bt**3/3 for all section elements

5.3 Lateral-Torsional Buckling of Beams

The design buckling resistance of a laterally unrestrained beam is evaluated in accordance with Clause 6.3.2.

$Mv = \chi_{LT}.Wpl.fy / mo$ Eq 6.54

χ_{LT} is the buckling resistance reduction factor Eq 6.56 or 6.57

For I & H sections Clause 6.3.2.3 is applied (λ_{LT0}=0.4, β=.75) for all other sections types 6.3.2.2 is applied. Note that 6.3.2.2 can be applied to I & H section by explicitly defining the imperfection factor as a negative value.

Program selected Imperfection factors are in accordance with Table 6.5 for I sections and Table 6.4 for other sections

χ_{LT} is can be evaluated using Mcr (Elastic critical moment) or the equivalent slenderness (λ_{LT}).

The equivalence of these two approaches is given by:

Non dimensional slenderness $\lambda_{bar} = (\lambda_{LT} / \lambda_1)(\beta w)^{0.5}$ where $\lambda_1 = 93.9\varepsilon$ and $\varepsilon = (235/fy)^{0.5}$

and

$Mcr = Wpl.fy / \lambda_{bar}^2$

The program using the following commonly used formula to evaluate Mcr for doubly symmetric sections (H & I sections).

$$Mcr = C1.\pi^2 E.Iz/L^2.\{[(k/kz)^2 Iw/Iz + (k.L)^2.G.It/(\pi^2.E.Iz) + (C2.z_g)]^{0.5} + C2.z_g\}$$

where k/kz = 1.0

It = Torsional constant (J in UK property tables)

Iz = Second moment of area (minor axis)

Iw = Wraping contant (H in UK property tables)

k.L = The effective length between lateral restraint

z_g = The distance between the point of load application and the shear centre

C1 = Moment distribution coefficient

C2 = Moment distribution coefficient

G = E*0.3846

To allow more accurate values of Mcr to be used, user defined values for Mcr can be specified.

The reduction factor f in Clause 6.3.2.3- Eq 6.58 is based on kc. kc is a user defined quantity (see section 4.2), the default value is 1.

The effective length (k.L) between lateral restraints and the moment loading condition factor are required to be defined by the user . The default value is the element length. This may not be conservative if the actual span length is longer than the element length and therefore should always be checked to ensure this reflects the restraint condition of the structure.

For I & H Sections Eqn 6.57 is applied using the buckling curves given in Table 6.5, b or c for rolled sections and c or d for welded sections. For all other sections Eqn 6.56 is applied using the 'd' buckling curve. The user can redefine the curves used by explicitly defining the Imperfection Factor.

The modified reduction factor χ_{LTmod} (Eqn 6.58) is evaluated using kc

The Equivalent Slenderness (λ_{LT}) approach to evaluate Mcr is is used for the following section types. Alternatively user defined values of Mcr may be specified.

Equivalent Slenderness (λ_{LT})

Note that when using the BS5950 correlations the equivalent uniform moment factor m_{LT} is applied as $1/C1^{0.5}$

I & H Section -Doubly Symmetric (Optional C2 = -1 activates option)

Uses BS5950(2000) Annex B B.2.3

I & H Sections Unequal Flanges

Uses BS5950(2000) Annex B B.2.3 (Small flange in compression)

Channel sections

Uses BS5950(2000) Annex B B.2.3

For Tee sections

Uses BS5950(2000) Annex B B.2.8 (Flange in tension)

For Box sections

Uses BS5950(2000) Annex B B.2.6

For L angle sections

Uses BS5950(2000) Annex B B.2.9 - Principle Axis Used

For Rectangular bar

Uses BS5950(2000) Annex B B,2.7

5.4 Tensile Load Capacities

The axial of members under tensile loading is in accordance with Clause 6.2.3. The smaller of ;

Npl = Afy / mo or Nu = 0.9.Ae.fu / m2

The user defines the effective area in terms of area ratio between effective area and gross area ie Ae/A.. The program default value is unity. This can be changed and the value saved.

5.5 Compressive Load Capacities

The axial resistance of members under compressive loading is in accordance with Clause 6.3 for flexural buckling (Cl 6.3.1.3). Torsional and torsional-flexural bucking checks are not undertaken (Cl 6.3.1.4).

Npl = Afy / mo Class 1, 2 or 3 cross sections

Nu = Ae.fy / m1 Class 4 cross sections

The compressive buckling resistance is in accordance with Clause 6.3.1.2

Nb = RF..A.fy / m1

RF is the reduction factor for strut buckling

The RF is a function of the slenderness ratio and the type of strut. The buckling curve is in accordance with Table 6.2. S460 curves are not defined. All non-welded box sections are assumed to be hot rolled (a - curve). The user can redefine the curves used by explicitly defining the Imperfection Factors.

The compressive strength is evaluated for both axes of the element and the lower value applied.

The slenderness ratio is dependent upon the effective length of the strut (kl). The program uses the length of the element as the default length and k = 1.0. The user may redefine the effective length and k value 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.

For L angles the buckling length (Lvv) for the principle axis is taken as the larger of the kl values for the geometric axis. For A type angles the slenderness is established in accordance with BB.1.2 and Lbb is used to define Lvv.

5.6 Combined Load Capacities - Strength

The load capacities (strength checks) of elements subject to combined loading are established in accordance with Clauses 6.2.8, 6.2.9 and 6.2.10.

5.6.1 Bending , Axial & Shear Force

With Low Shear

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 may be used for Class 1 and Class 2 sections (Clause 6.2.9).

The following are used to evaluate the reduced moment resistance.

Section Type	Clause	Equation
I or H	6.2.9(5)	6.36, 6.37 & 6.38
Boxes	6.2.9(5)	6.39 & 6.40
Rect Bar	6.2.9(3)	6.32

For CHS sections the reduced plastic moment capacity is based on Eqn 5.34 from ENV 1993-1-1:1992. Mnrd =1.04.Mpl.(1-$n^{1.7}$) where n=Nsd/Npl

Bi-axial bending is accordance with Equation 6.41 for sections that use the reduced moment capacity. If the TI parameter is set to 1 all checks combined checks will use Eqn 6.1.

To provide more compatibility with ISO 19902 the CHS combined load load check can be based based on Eqn 6.2 but using $\sqrt{(M_y^2 + M_z^2)}$. To do this the global TI parameter should be set to the value 1.

Combined loading on other Class 1, Class 2 and Class 3 sections use only the simplified axial/bending interaction formula (Clause 6.2.1(7) Eqn. 6.2) and do not use a reduced moment capacity.

For Class 4 sections equation 6.2 is also used since the conservative approach using a strength reduction factors for slender sections does not effect section properties (Sect 5.1.3), effectively equation 6.44 is reduces to the same form as 6.2.

With High Shear

In accordance with Clause 6.2.10 the effect of shear is included by using the simplified interaction formula (Eq 6.2) for Class 1 & 2 sections in which the component resistances are reduced due the presence of shear. Class 3 & 4 sections also use equation 6.2 in which the component resistances are reduced due the presence of shear.

5.7 Combined Load Capacities - Buckling

In addition to the combined strength check the following combined buckling checks are undertaken for members subjected to axial compression.

5.7.1 Compression Members with Moments

The design buckling resistance for members subjected to combined bending and axial compression is in accordance with Clause 6.3.3 using equations 6.61 and 6.62.

The interaction factors are evaluated using the Annex B method.

The Cmx, Cmz and CmLT factors which are a function of loading type and end support conditions are user defined quantities. The program default values are unity. These values can be modified and saved in a result case associated data file.

This check is applied to all section types.

-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

Data Files and Model Files

"modelname".ELN	Modified Element Parameter File
"modelname".EDP	Modified Geometric Property File
"modelname".^E"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

Modified Geometric Properties File Format

The .EDP 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 **DP**. All data must be present on the line. The format for the line is shown below.

DP, *GCode, ARatio, GirderStiffSpace, ImpFact-x, ImpFact-x, ImpFact-Lat*

GCode	Property Code Number
ARatio	Area ratio fro tensile capacities
GirderStiffSpace	Distance between plate girder stiffeners
ImpFact-x	Column buckling curve inperfaction factor x-x axis .
ImpFact-x	Column buckling curve inperfaction factor y-y axis
ImpFact-Lat	Lateral buckling curve inperfaction factor

-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.

EC3 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(Setting to -3 will activate 21pt check).

C4 Not used

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-