FS2000

AISC 360 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 AISC360 CodeCheck is an interactive program module the interfaces with FS2000 to provide design checks in accordance with specific requirements of The American Institute of Steel Construction (AISC) "The AISC Specification for Structural Steel Buildings" AISC 360-16 Edition.

The codechecker will check the following structural sections

Symmetrical I sections

Channel sections

Rectangular hollow sections (boxes)

Circular hollow sections

Structural Angles (strut/tie only - A Angles)

Structural angles (with bending -L-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

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

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

SI System

Length	metres (m)
Force	Newton (N)
Moment	Newton-metre (Nm)
Stress	N/m2(typically for steel E=205E9 Grade 50 Yield=345E6)

USA System

Length	inches (ins)
Force	pound (Lbs)
Moment	pound-inch (Lb-ins)
Stress	Lbs/ins2 (psi) (typically for steel E=30E6 Grade 50 Yield=50E3)

-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, and

2 Element steel grades

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

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

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

Angle Sections

FS2000 has two types of angle sections.

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

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

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

-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

Approximate Second Order Analysis

CMx *Moment reduction factor*

CMy *Moment reduction factor*

Lateral-Torsional /Buckling

Llb Unsupported Length of Compression Flange

Cb *Lateral buckling parameter*

Plate Girder Design

Stiffener spacing

Buckling Lengths

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

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

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

Moment Factors

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

-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.

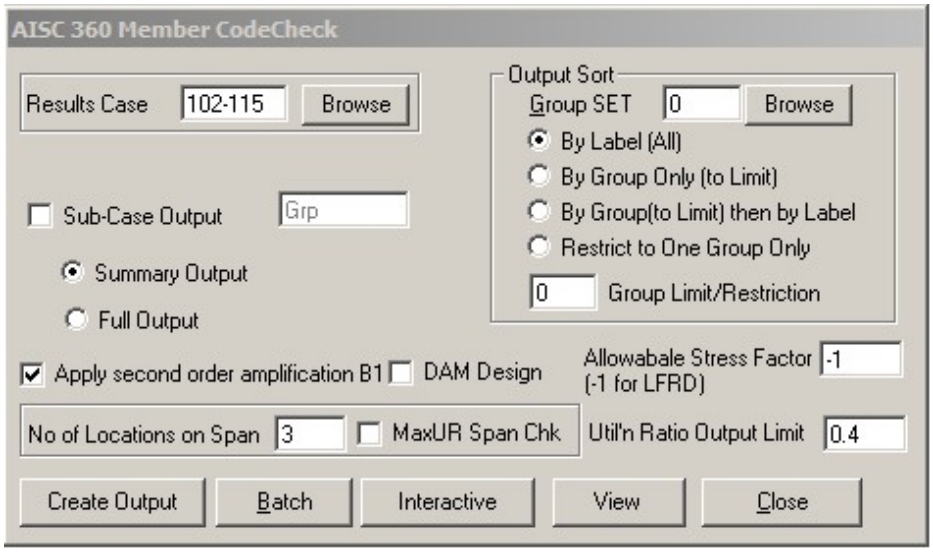
The text out from the codechecker can be view one of the following.

- 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

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



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

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

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

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

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

Interactive Mode option activates Interactive Operation of the CodeChecker

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

The **Apply second order amplification B1** option can be used to deactivate second order amplification for P-delta (P- δ) effects between brace points (Clause Eq A-8-3). Only to be de-activated if non-linear P-Delta solution is undertaken with an adequate number of mid span nodes. See Section 5.1.

The **Dam Design** option is used to identify that the case being checked was a solution based on 80% of the nominal stiffness (Clause C2.3) See Section 5.1.

Allowable Stress Factor If this value is set as -1 then this will indicate that the design check is to be an LFRD check using a factored result case. If this value is set as 1 then this will indicate that the design check is to be an ASD check using a non factored result case. Non-unity values may be defined. If a value of -1.058 is set for the LRFD format the ϕ (0.9) value will be factored by 1.058 (useful for applying the Ks factor from ISO 19902-3). If a value of 1.333 is set for the ASD format then this will be interpreted as a factor to increase the allowable yield stress of an ASD design check. Note that this stress increase may not increase the allowable load levels on buckling related limits.

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.

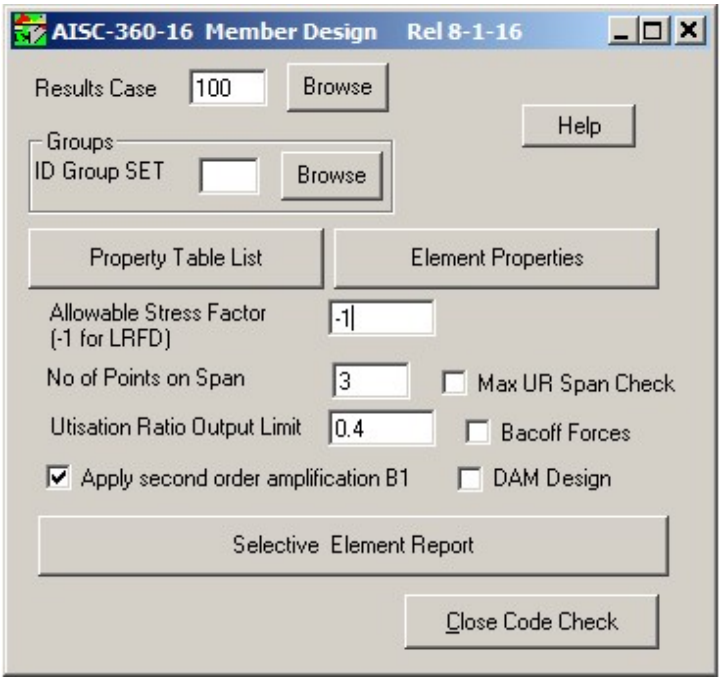
Important Note Stress ratio data created by this module will be limited to the data processed. If restricted process options is used then any Stress Ratio plots or Stress Ratio sorts which use the same results cases will be limited to the processed data. The plot or listed output will indicate if the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

-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 AISC 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 **Apply second order amplification B1** option can be used to deactivate second order amplification for P-delta (P- δ) effects between brace points (Clause Eq A-8-3). Only to be de-activated if non-linear P-Delta solution is undertaken with an adequate number of mid span nodes. See Section 5.1.

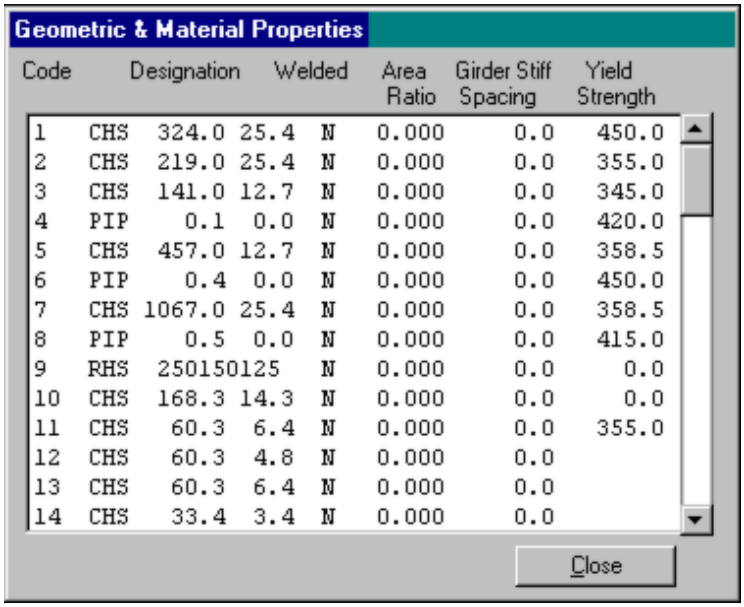
The **Dam Design** option is used to identify that the case being checked was a solution based on 80% of the nominal stiffness (Clause C2.3) See Section 5.1.

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-

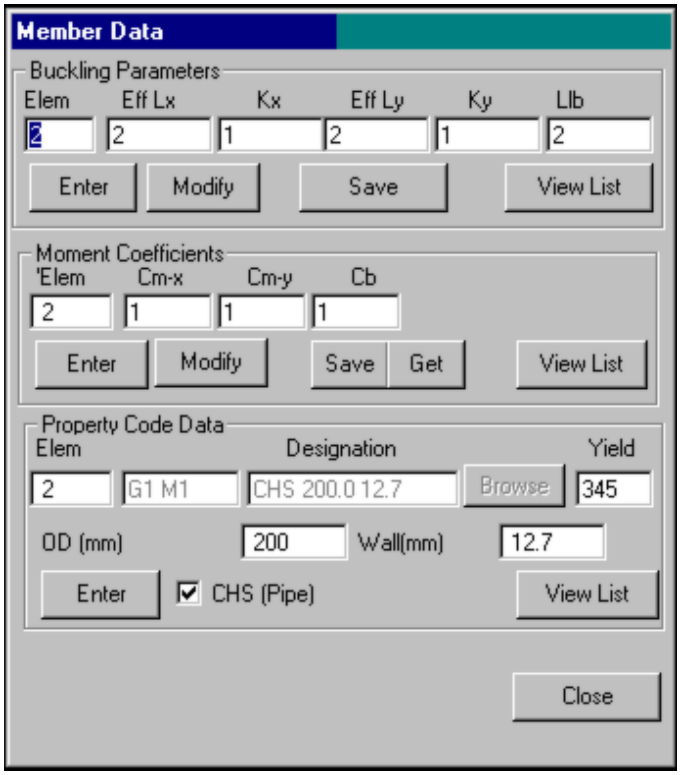
4.0 Modification of Element Properties

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

UNITS

SI-Units length in metres

USA-Units length in inches

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.

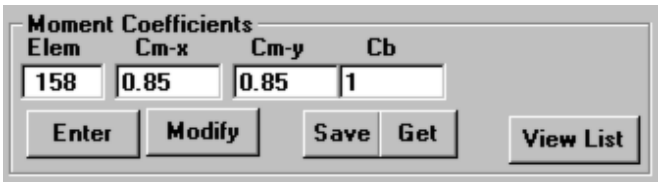
The **Save** button is used to save all current element buckling data entered to the definition file (.ELN file). This is a formatted text file which should be included in any analysis report. If one exists it will always be loaded when the code check module is started. The file may be edited. The instruction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

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 .^'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.

Approximate Second Order Analysis - Appendix 8

CMx Moment reduction factor

CMy Moment reduction factor

Extract from AISC

C_m = coefficient assuming no lateral translation of the frame determined as follows:

(a) For *beam-columns* not subject to transverse loading between supports in the plane of bending

$$C_m = 0.6 - 0.4(M_1/M_2) \qquad \text{(A-8-4)}$$

where M_1 and M_2, calculated from a *first-order analysis*, are the smaller and larger moments, respectively, at the ends of that portion of the member unbraced in the plane of bending under consideration. M_1/M_2 is positive when the member is bent in *reverse curvature*, negative when bent in *single curvature*.

(b) For beam-columns subject to transverse loading between supports, the value of C_m shall be determined either by analysis or conservatively taken as 1.0 for all cases.

Lateral Torsional Buckling

Cb Lateral buckling parameter

Extract from AISC

C_b, the *lateral-torsional buckling* modification factor for nonuniform moment diagrams when both ends of the segment are braced is determined as follows:

$$C_b = \frac{12.5M_{max}}{2.5M_{max} + 3M_A + 4M_B + 3M_C} \qquad \text{(F1-1)}$$

where

M_{max} = absolute value of maximum moment in the unbraced segment, kip-in. (N-mm)

M_A = absolute value of moment at quarter point of the unbraced segment, kip-in. (N-mm)

M_B = absolute value of moment at centerline of the unbraced segment, kip-in. (N-mm)

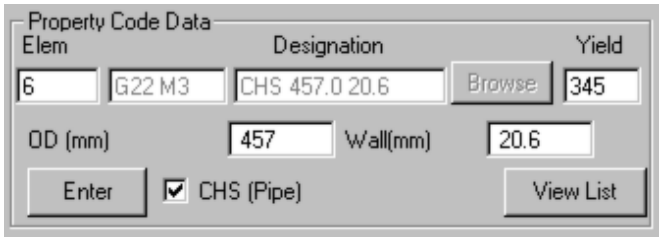
M_C = absolute value of moment at three-quarter point of the unbraced segment, kip-in. (N-mm)

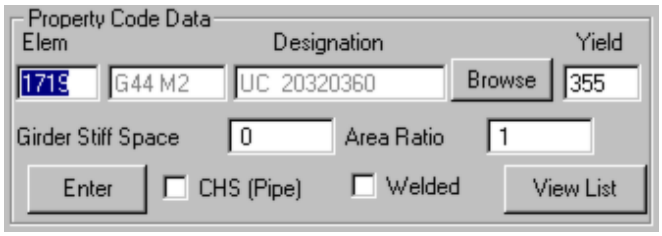
For cantilevers or overhangs where the free end is unbraced, $C_b = 1.0$.

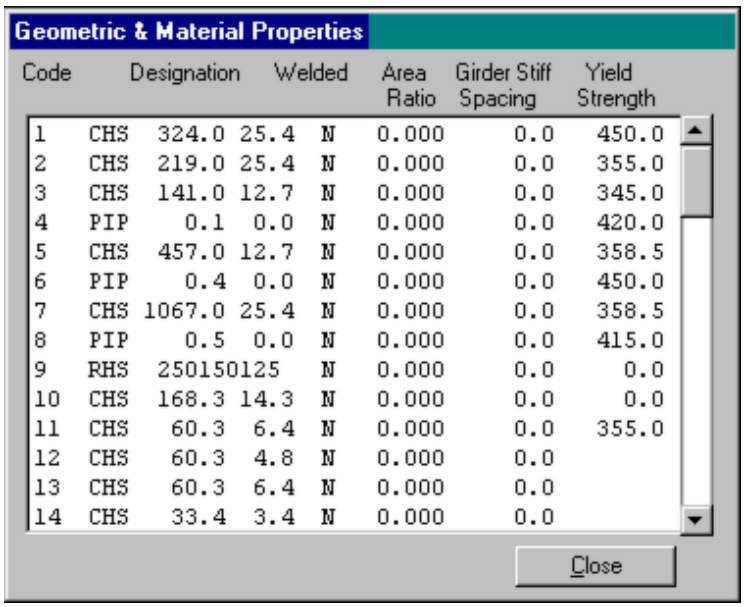
-o-

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 **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 **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.

-o-

4.4 Buckling Parameters - Graphical Definition

The 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 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 AISC are used by the program and how they are interpreted.

5.1 Design Strength & Stability

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

Stability factors B1 & B2 (Appendix 8)) are employed to assess stability based on first order elastic analysis. The B2 factor accounts for the second order effects due to the displacement of braced points. This B2 factor is not included in the FS-AISC code checker. The B1 factor, used to account for displacements between brace points, is evaluated using the Cm constants (see Section 4.2). Setting the **Member 2nd Order** check box to unchecked will set B1=1 for all elements and remove this check. This may be done in cases where P-Delta analysis is undertaken and sufficient mid span nodes are present to account for mid span deflections.

5.2 **Axial Tension**

All sections are checked in accordance with AISC clause D2 using the lower of the design strengths defined by D2-1 or D2-2.

The Area Ratio which is a property code attribute is used to define the effective area of the section.

If the ultimate tensile strength is not defined the design strength is based on the material yield strength using only D2-1

5.3 **Axial Compression**

The effective slenderness is taken as the largest of (KL/r)x or (KL/r)y. In the case of L-Angles the principal axis slenderness are also included.

Flexural Buckling (non-slender) Clause E3

All sections but excluding A-Angles	E3-1 to E3-4

Torsion and Flexural-Torsional Buckling (non-slender) Clause E4

I Beams	E4.1 and E4-2
Channels	E4.1 and E4-3
L-Angles	E4.1 and E4-4
Tees	E4.1 and E4-2

In E4-4 KzL is taken as Llb.

In E4-9KzL is taken as Llb.

In E-4-9 Cw is taken as zero for L-Angles

Single Angles (A-Angles) (non-slender) Clause E5

The slenderness is taken as the largest of Lx and Ly and the r is taken to be ry.

The effective KL/r cannot be less that KL/rz

The effective slenderness evaluated using E5-1 or E5-2 as appropriate.

Members with Slender Elements Clause E7

For all section with elements that are classified as slender (Table B4.1), Clause E7-1 is used to establish the compressive strength. The slender reduction factor Q is evaluated using the following clauses. The capacity is based on the lowest Q for the section.

I Beam & Channel flanges (Rolled)	E7-4, E7-5 and E7-6
I Beam & Channel flanges (built-up)	E7-7, E7-8 and E7-9

I Beam & Channel webs	E7-16 & E7-17
Tee flanges (Rolled)	E7-4, E7-5 and E7-6
Tee (built-up)	E7-7, E7-8 and E7-9
Tee stems	E7-13, E7-14 and E7-15
A-Angles and L-Angles	E7-10, E7-11 and E7-12
CHS	E7-19
Box sections	E7-16 & E7-18

5.4 **Bending**

5.4.1 ***IBeams - Major Axis Bending***

Yielding	F2-1
Lateral torsional buckling	(LTB) F2-2 or F2-3
Flange local buckling (FLB)	F3-1 or F3-2
Noncompact webs -yielding	F4-1
Noncompact webs -LTB	F4-2 or F4-3
Noncompact webs -FLB	F4-12 or F4-13
Slender webs -yielding	F5-1
Slender webs -LTB	F5-2
Slender webs -FLB	F5-8 or F5-9

IBeams - Minor Axis Bending

Yielding	F6-1
Flange local buckling (FLB)	F6-2 or F6-3

Channels - Major Axis Bending

Yielding	F2-1
Lateral torsional buckling	F2-2 or F2-3
Flange local buckling (FLB)*	F3-1 or F3-2
NC or Slender webs -yielding*	F5-1
NC or Slender webs -LTB*	F5-2
Nc or Slender webs -FLB*	F5-8 or F5-9

Channels - Minor Axis Bending

Yielding	F6-1
Flange local buckling (FLB)	F6-2 or F6-3

Tees- Major Axis Bending

Yielding	F9-1
Lateral torsional buckling	F9-4 (B is always negative - Stem in compression)
Flange local buckling (FLB)	F9-6 or F9-7
Stem local buckling	F9-8, F9-9 or F9.10 or F9-11

Tees - Minor Axis Bending

Yielding	F6-1

Flange local buckling (FLB)	F6-2 or F6-3

L-Angles

Bending is based on the principle axis . Moments are transformed from the geometric axis to the principle axis. The output only shows the principle axis values, x-x and y-y should be interpreted as u-u and v-v.

Yielding	F10-1
Lateral torsional buckling	F10-2 or F10-3
Equal	F10-5
Unequal	F10--6
Leg local buckling	F10-6 or F10-7

Boxes - Major and Minor Axis Bending

Yielding	F7-1
Flange local buckling	F7-2 or F7-3
Web local buckling	F7-4

CHS

Yielding	F8-1
Local buckling	F8-2 or F8-3

Rectangular and Round Bars

Yielding	F11-1
Lateral torsional buckling	F11-2 or F11-3

5.5 **Shear**

IBeams - Fy In plane of web

Nominal G2	G2-1
Rolled G2.1.(a)	G2-2
Built-up (Deep rolled)	G2-3, G2-4 or G2-5
(If a<h a=h)	

IBeams - Fx in plane of flanges

Nominal G7	G2-1
With kv = 1.2	G2-3, G2-4 or G2-5

Channels - Fy In plane of web

Nominal G2	G2-1
Built-up (Deep rolled)	G2-3, G2-4 or G2-5
(If a<h a=h)	

Channels - Fx in plane of flanges

Nominal G7	G2-1
With kv = 1.2	G2-3, G2-4 or G2-5

Boxes

Nominal G5	G2-1
With kv as for plate buckling	G2-3, G2-4 or G2-5

Full width ????

CHS

Nominal	G6-1
Fcr	G6-2b

Tees

Nominal	G2-1
With kv = 1.2	G2-3, G2-4 or G2-5

L-Angles

Nominal	G2-1
Cv =1	G4

Rectangular Bars

Nominal G7	G2-1
With kv = 1.2	G2-3, G2-4 or G2-5

5.6 **Torsion**

CHS

Nominal	H3-1
Fcr	H3-2a or H3-2b

Boxes

Nominal	H3-1
Fcr	H3-3, H3-4 or H3-5

Other Sections

Nominal	H3-8

The geometric property code Torsional Stress Modulus is used to calculate the shear stress.

5.7 **Combined Loading**

Second order effects are using the B1 moment factor A-8-3.

IBeams, Channels, Boxes & CHS

Combine UR	H1-1a or H1-1b

Tees, L-Angles & Bars

Combine UR	H2-1

Boxes & CHS with Torsion (Tr/Tc>0.2)

Combined UR	H3-6

For Boxes

Mr/Mc is taken as Mrx/Mcx + Mry/Mcy in H3-6

Vr/Vc is taken as the larger of Vx/Vcx or Vy/Vcy in H3-6

For CHS

Mr/Mc is taken as ((Mrx/Mcx)**2 + (Mry/Mcy)**2)**.5 in H3-6

Vr/Vc is taken as ((Vx/Vcx)**2 + (Vy/Vcy)**2)**.5 in H3-6

-o-

5.1 Stability

Stability requirements in AISC are required to take the following into account: Member deformations, Second order effect (P-Delta), Geometric Imperfections, and Stiffness Reduction.

AISC identifies two distinct second order effects:
P-Delta (P-Δ) and P-delta (P-δ). P-Delta is global sway e.g. story drift and P-delta is member curvature.

Geometric imperfection cans be included in the model geometry or applied as notional loads (0.002)

Two basic approaches can be employed to achieve this:

- Direct Analysis Method DAM

  The stiffness of the solution model is to be based on 80% of the nominal stiffness.

  In this method the solution has to take into account both P-Delta (P-Δ) and P-delta (P-δ). This can be accomplished using either:
  1. A model with a segmented members such the the effects of P-delta (P-δ) is included. This it is recommended that at least 3 mid nodes be used for non-sway frames and 1 for sway-frames.
  2. A model where only P-Delta (P-Δ) effects are included (no mid-span nodes) and the P-delta (P-δ) effects are included in the design checks using a moment amplification (B1 in A-8-3)

  The design checks have to be under taken using the full member stiffness. Effective length factor K=1 for all elements

  The DAM option is used to activate the factor the EI stiffness (A-8-3) for the code check. Note that τ is a always 1 and thus assumes that a notional load (0.001) is applied (Clause C2-3c).

- Effective Length Method ELM

  The stiffness of the model is to be based on 100% of the nominal stiffness.
  Effective length factors (K) have to be defined and moment amplification included.

From the above it can be concluded that the DAM method is very easily implemented using FS2000 using the following steps.

1. Create a model using the the nominal E value. Additional mid span nodes are not required if the B1 amplification factor is to be used.
2. Re-save the model with reduced E values (80%)
3. Under take the P-Delta second order solution (with the reduced stiffness).
4. Re-save the mode with the full E value (100%) using K=1 for all members
5. Undertake the AISC design checks

At Step 4 the model has to be re-saved. When saving the model in interactive mode the Purge Results option must be in-active, otherwise the reduced stiffness results will be deleted.
When running in Batch mode the model stiffness before and after solution would be modified using Dynamic Interpretation.

-o-

APPENDIX A - Geometric Property Libraries

Geometric Property Libraries

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

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

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

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

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

Property File	Type	Description
UB.PRI	I	Universal beams
UC.PRI	I	Universal columns
RSJ.PRI	I	Joist sections
PFC.PRC	C	Channel sections (RSC also)
RHS.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

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

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

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

WS_.PRI	I	W Shapes

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

Creating and Editing Property Libraries

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

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

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

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

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

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

Model Dependent Property Libraries

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

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

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

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

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

Geometric Library File Formats

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

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

```
INCH   ASTM  W Shapes
Desig  D    B    t    T    r     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.

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
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
```

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
```

Welded Sections

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

-o-

APPENDIX B - Data Files

Data Files and Model Files

Files created by the code checker are:

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

"n" is the Results Case Number

Modified Parameter File Format

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

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

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.

AISC5 C1/C2/C3/C4/C5/C6/C7/C8/

C1 The processed results case number. Ranges may be processed e.g. 1-10 would process Results Cases 1 to 10 . If cases do not exist when processing ranges a warning will be given and the process will continue to the next case.

C2 Text file output format 2-Summary Report, 3-Full Report

C3 Number of location on span for code check (Setting to -3 will activate 21pt check)

C4 Allowable stress increase factor.

C5 Unity check ratio limit for output.

C6 Group SET to read

C7 Group Limit/Restriction

C8 Groups Only switch

C9 Subcase name

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.

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

-o-