FS2000

AISC 9th Ed 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 AISCCodeCheck is an interactive program module the interfaces with FS2000 to provide design checks in accordance with the code requirements of The American Institute of Steel Construction (AISC) "The AISC Specification for Structural Steel Buildings" Allowable Stress Design (ASD) - 9th Edition Manual. Circular hollow sections are checked in accordance with requirements of API Recommended Practice for Planning, Designing and Constructing Fixed Offshore Platforms (RP2A 21th Edition 2005 Supp) by the American Petroleum Institute.

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

The AISC codechecker will check the following structural sections

Symmetrical I sections

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.

Tapered sections

-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. It is also used to combined multiple results cases in to single processed results using pre-defined load case combinations.

-o-

2.3 Property Codes

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

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

For a section property to be checked it must 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-

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

Tensile Strength

Area Ratio Defines the effective area (tensile capacities).

Column Buckling

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

Kx Effective length for xx Comp buckling

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

Ky Effective length for yy Comp buckling

Lateral-Torsional /Buckling

Llb Unsupported Length of Compression Flange

CMx *Moment reduction factor*

CMy *Moment reduction factor*

Cb *Lateral buckling parameter*

Plate Girder Design

Stiffener spacing

Buckling Lengths

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

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

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

Moment Factors

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

-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.

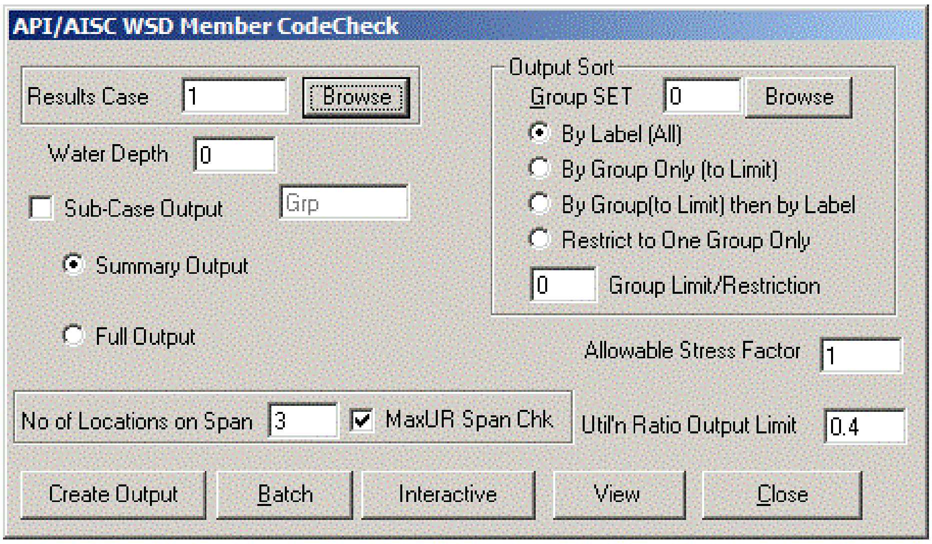
The text out from the codechecker can be view one of the following.

- 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.

The **Water Depth** is used to define the depth of water of the the combined hydrostatic loading checks. This has two modes of operation depending whether the external pressure is define explicitly in a load case or is not (see Section 4.0). If **Llb** is defined and not equal to the element length, **Llb** will used as the cylinder length for the buckling checks, otherwise the maximum effective length will be used (larger of **EffLx** or **EffLy**).

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

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

Interactive Mode option activates Interactive Operation of the CodeChecker

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

Allowable Stress Factor is used to enter a factor to increase the allowable stress i.e. the 1/3 increase commonly used for environmental loading would be input as 1.333.

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

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

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

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

Groups

The **Group SET** box is used to define the group SET to be loaded. If a SET is loaded then all node and element labels will be accompanied by their respective group attribute. If this field is left blank or contains the number of a non existent group then only the basic node and element numbers will be used for reference in the lists.

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

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

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

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

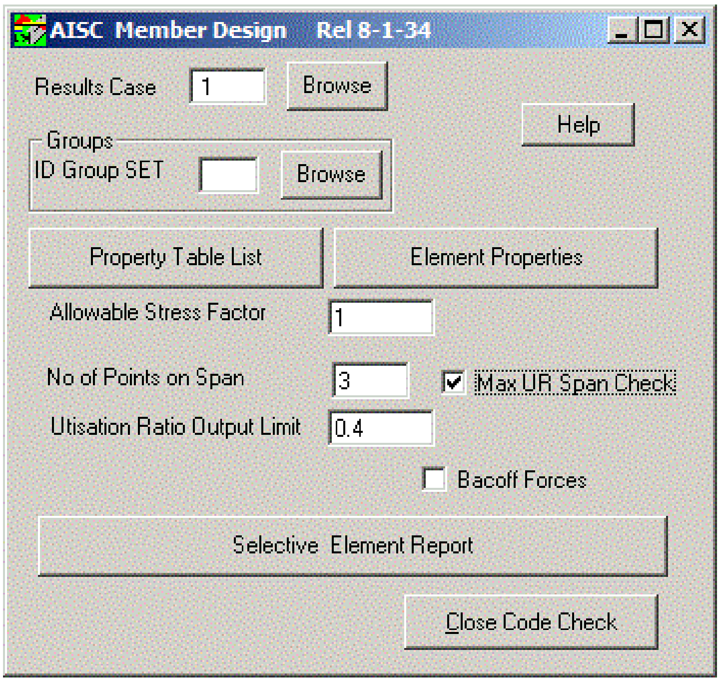
Important Note Stress ratio data created by this module will be limited to the data processed. If restricted process options is used then any Stress Ratio plots or Stress Ratio sorts which use the same results cases will be limited to the processed data. The plot or listed output will indicate if the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

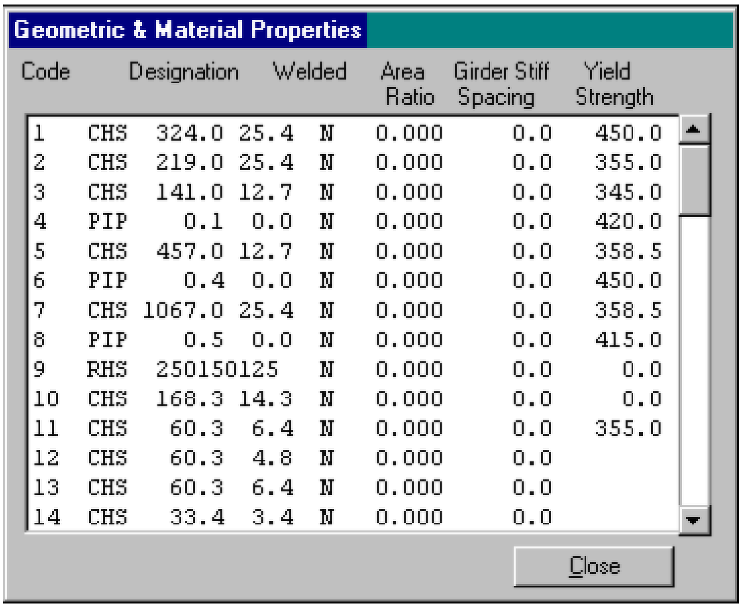
The **Element Properties** button is used to view and define elements properties. Re-defining properties enables the user to design the element for the applied loading. The re-definition of section properties is described in Section 4.0.

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-

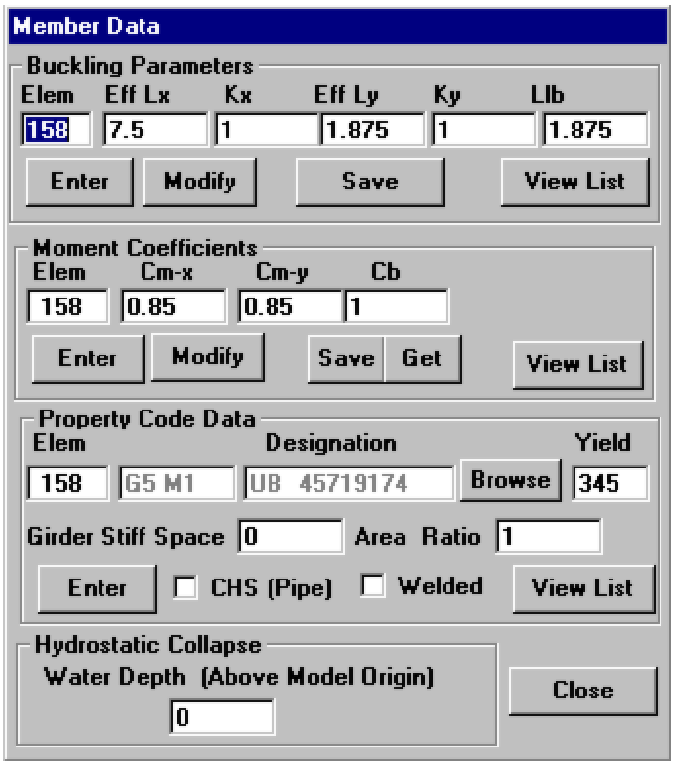
4.0 Modification of Element Properties

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



The **Water Depth** is used to define the pressure for the combined hydrostatic loading checks. This has two modes of operation depending whether the external pressure is define explicitly in a load case or is not.

- Explicit External Pressure Definition

  If the external pressure is defined explicitly in a load case or in a load case generated from FS-Wave the the **Water Depth** parameter is interpreted only as a switch. Any non-zero value will active the collapse check based on the result case internal and external pressure i.e. Hoop is based on Pi - Po. If Pi > Po Hoop =0.

- Non-Explicit External Pressure Definition

  The **Water Depth** is used to define the depth of water of the combined hydrostatic loading checks. If a non zero value is entered and Po is zero this value will be interpreted as the depth of water relative to the model Y origin. A negative value would imply that the surface elevation is below the origin. The local external pressure is evaluated based on a water density of 1025 kg/m3. Hoop is based only on the water depth i.e. Pi = 0.

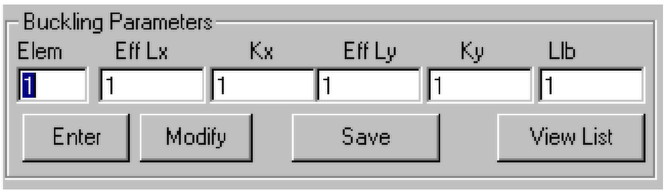
If **Llb** is defined and not equal to the element length, **Llb** will used as the cylinder length for the buckling checks, otherwise the maximum effective length will be used (larger of **EffLx** or **EffLy**).

.

-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

-Also applies to Depth for hydro checks

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

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

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

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

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

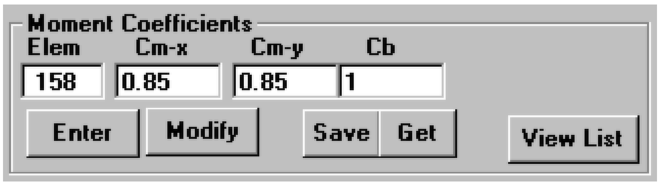
The **Save** button is used to save all current element buckling data entered to the definition file (.ELN file). This is a formatted text file which should be included in any analysis report. If one exists it will always be loaded when the code check module is started. The file may be edited. The instruction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

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

-o-

4.2 Moment Coefficients

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



CMx Moment reduction factor

CMy Moment reduction factor

Cb Lateral buckling parameter

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

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

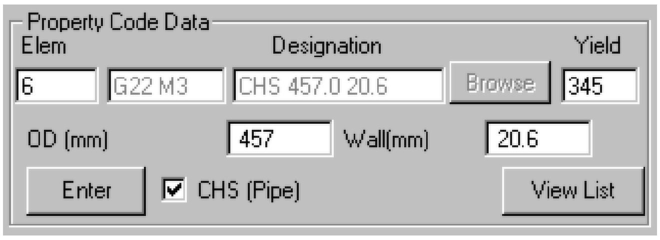
The **Get** button is used to retrieve a previously saved moment factor file. Always ensure that the retrieved case is related to the current load case file.

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

-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.



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

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

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

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

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

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

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

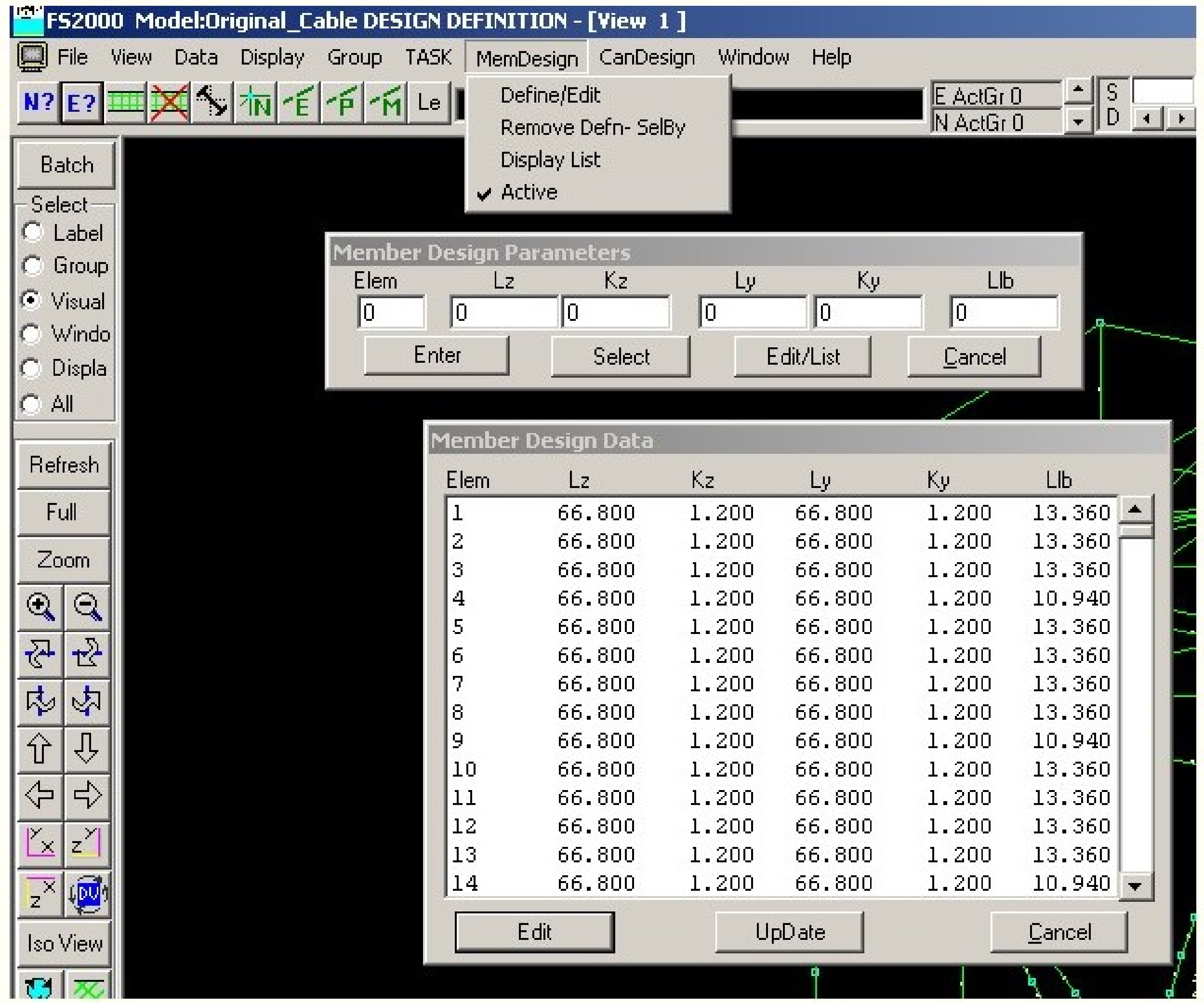
The **CHS** check box is used to change the property code to that for a pipe. When checked, input boxes to for the OD and wall thickness of a tube will appear. The Enter button is used to enter the values displayed in the boxes. Use the **ViewList** to check the entries.

-o-

4.4 Buckling Parameters - Graphical Definition

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

This facility is available in the Design TASK.



-o-

5.0 Technical Specification

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

5.1 Design Strength

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

5.2 **Axial Tension**

Tubular sections are checked in accordance with API Eq 3.2.1-1

Ft = 0.6Fy

Other sections are also checked to the same criteria in accordance with AISC clause D1. The tensile stress is evaluated using the effective area ratio.

5.3 **Axial Compression**

Both tubular and non-tubular sections are checked in accordance with the AISC column buckling criteria clause E2.

(1- (Kl/r)2/2Cc2)Fy

Fa = -- E2-1

5/3 + 3(Kl/r)/8Cc - (Kl/r)3/8Cc3

Cc = 12PI2E/23(Kl/r)2

If Kl/r > Cc thenFa = 12PI2E/23(Kl/r)2 E2-2

For tubes the local buckling stress limits in API Clause 3.2.2b are applied.

5.4 **Shear**

Beam Shear

Tubular member are checked in accordance with API Eqn 3.2.4-1

Fv = 0.4Fy

The shear stress is evaluated on the vector sum of the principal shears based on 0.5 the c.s.a.

Non-tubular sections are checked in accordance with F4

Fv = 0.4Fy

I sections and channels are also checked in accordance with AISC clause F4-2 for web shear.

F = Fy/2.89(Cv) =< .4Fy F4-2

Cv = 45000/Fy(h/t)2 when cv < 0.8

Cv = 190(k/Fy).5/(h/t)

k = 4 + 5.34/(a/h)2 for a/h < 1

k = 5.34 + 4/(a/h)2 for a.h > 1

Shear in the direction parallel to the flanges is based on 0.4Fy and is evaluated using the full area of both flanges.

Torsional Shear

Torsional shear stress in tubes are checked in accordance with API Eqn 3.2.4-3

fvt = Mt(D/2)/Ip Fvt = 0.4.Fy

Torsional shear stress in box section are checked by the following (not a code requirement)

fvt = Mt/Zt (Zt=2Tw(B-Tw)(D-Tf)) Fvt = 0.4.Fy

5.5 Bending

5.5.1 Bending Tubulars

The permissible bending stress in tubular section is established in accordance with API clause 3.2.3.

Fb = 0.75Fy for D/t =< 10340/Fy

Fb = (0.84-1.74FyD/Et)Fy for 10340/Fy < D/t =< 20680/Fy

Fb = (0.72-0.58FyD/Et)Fy for 20680/Fy < D/t =< 300

5.5.2 Bending Non-Tubulars - Non-Slender

The following compact limit checks from AISC clause B5.1 are applied.

a) Flanges bf /tf=< 65/(Fy).5

For box sections

bf /tf=< 190/(Fy).5

b) Webs For fa/Fy =< 0.16

d/t =< 640/Fy(1-3.74fa/Fy)

For fa > 0.16

d/t =< 257/(Fy).5

c) Comp Flange Stabilty (Lat Buck) Clause F1.1

Lc <= 76bf/Fy0.5 nor LC <= 20000/(d/Af)Fy

5.5.2.1 Major axis bending - I Section & Channels

If a), b) and c) above are satisfied ie compact then

Fbx = 0.66Fy F1-1

If b) and c) are satisfied and bf/2tf < 95/Fy.5 then

Fbx = FPy(0.79-.002(bf/2tf(Fy/kc).5) F1-3 & F1-4

If a) is not satisfied for Box sections then Fbx = 0.6Fy

5.5.2.2 Minor axis bending - I Sections & Channels

If a) above is satisfied then Fby = 0.75Fy F3-1

If 95/Fy.5 > bf/2tf > 65/Fy.5 then

Fbx = FPy(1.075-.005(bf/2tf)Fy.5) F3-2

If a) is not satisfied for Box sections then Fby = 0.6Fy F2-2

If a) is not satisfied for Channel sections then Fby = 0.6Fy

5.5.2.3 Major & minor axis bending - T Sections

Flanges If a) and b) above are satisfied ie compact then

Fbx = 0.66Fy F1-1

Else Fbx = 0.6Fy

Stem Fb = Flange Fb

5.5.2.4 Major and minor axis bending - Box sections

Flanges If a) above is satisfied ie compact then

Fb = 0.66Fy F3-1

Else Fb = 0.6F F3-3

5.5.2.5 Major and minor axis bending - Solid rectangular bar

Fb = 0.75Fy F2-1

5.5.2.6 Bending - angles

If a) above is satisfied ie compact then

Fb = 0.66Fy 5-1a

Else Fb = 0.6F 5-1c

5.5.2.7 Non Compact Tension Flanges

For any case not covered above and with the flanges in tension then

Fb = 0.6Fy

The flange is taken to be tensile only in cases where the axial tensile stress exceeds the bending stress i.e. the member is predominately a tension member.

5.5.2.8 Lateral/Torsional Buckling

Lateral/Torsional Buckling checks are only applied to I sections and channel sections. These checks are in accordance with section F1.

Compression Flange Stability (I Sections & Channels)

For any case not covered above and with the flange in compression and bf/2tf<95/Fy.5 then in accordance with F1.3 Fbx is taken as the larger of the following but not more than 0.6Fy .

when (102E3Cb/Fy).5 =< l/rT =< (510Cb/Fy).5

Fbx = (2/3-Fy(l/rT).5/1530E3Cb)Fy F1-6

where l/rT => (510Cb/Fy).5

Fbx = 170E3Cb/(l/rT).5 F1-7

or

Fbx = 12E3Cb/(ld/Af) F1-8

Equation F1-8 is the only one applied to Channels

5.5.3 Bending in Slender Sections

5.5.3.1 Bending with Thin Webs (Plate Girders - I section and Channels)

When d/t > 760/Fb2 then the compression flange bending stress is taken as

Fb' = Fb(1-.0005Aw/Af(h/t-760/Fb.5)) G2

5.5.3.2 Appendix B Considerations for Slender Sections

Qs is evaluated using the following for axially loaded members and flexural members containing elements subject to compression which have a width-thickness ratio in excess of the applicable non-compact value in Sect. B5.1.

The flange is taken to be tensile only in cases where the axial tensile stress exceeds the bending stress i.e. the member is predominately a tension member. In such case Qs=1

Fb =Qs.0.6Fy but not > 0.6Fy

Qs is applied to axial compression in accordance with B5.2.c

I Sections and Channels

When $b_f/2t_f > 95/F_y^{.5}$ then the reduction factor from Appendix B is employed. This is only applicable to I & H sections.

When $95/(F_y/k_c)^{.5} < b/t < 195/(F_y/k_c)^{.5}$

$Q_s = 1.415 - .00437(b/t)(F_y/k_c)^{.5}$ A-B5-3

when $b/t \Rightarrow 195/(F_y/k_c)^{.5}$

$Q_s = 26200k_c/(F_y(b/t)^2)$ A-B5-4

Max allowable bending stress = $0.6F_yQ_s$ for both major and minor axis bending.

The above slenderness reduction factors are also applied to the member compression capacity in accordance with Clause B5.2.c.

Box Sections

Clause B5.2.b for box sections is not employed but a strength reduction is applied if sections are slender due to flange/web proportions ie $b_f/t_f > 238/F_y^{.5}$. The Q_s strength reduction factor applied is

$Q_s = (238/F_y^{.5} / b_f/t_f)^2$

This will be conservative compared to the Clause5.2.b method

Tee Sections

The Qs applied is the lower of the flange and stem for major axis bending and axial compression. Minor axis bending is based on the flange Qs

Flanges

When $95/(F_y/k_c)^{.5} < b/t < 195/(F_y/k_c)^{.5}$

$Q_s = 1.415 - .00437(b/t)(F_y/k_c)^{.5}$ A-B5-3

when $b/t \Rightarrow 195/(F_y/k_c)^{.5}$

$Q_s = 26200k_c/(F_y(b/t)^2)$ A-B5-4

Stems

When $127/(F_y/k_c)^{.5} < b/t < 176/(F_y)^{.5}$

$Q_s = 1.908 - .00715(b/t)(F_y)^{.5}$ A-B5-5

when $b/t \Rightarrow 175/(F_y/k_c)^{.5}$

$Q_s = 20000k_c/(F_y(b/t)^2)$ A-B5-6

Angle Sections

The Qs applied is the lower of the two legs and is applied to both axis of bending and axial compression.

When $76/(F_y/k_c)^{.5} < b/t < 155/(F_y/k_c)^{.5}$

$Q_s = 1.34 - .00447(b/t)(F_y)^{.5}$ A-B5-1

when $b/t \Rightarrow 155/(F_y/)^{.5}$

$Q_s = 15500k_c/(F_y(b/t)^2)$ A-B5-2

Solid Rectangular bar

No slenderness checks are applied to this type of section

Stiffened Elements

Sections that are slender due stiffened elements may subjected to a reduction factor (Qa, see B5.2.c) for axial load compression. This reduction is not evaluated.

5.6 Combined Loading

5.7.0 Combined Bending in Tubulars

Axial Compression + Bending

The maximum permissible combined loads are checked against the requirements of API clause 3.3.1.

For fa/Fa > 0.15

fa/Fa + ((Cmxfbx/(1-fa/F'ex)2+(Cmyfby/(1-fa/F'ey)2).5/Fb   3.3.1-4

fa/0.6Fy + (fbx2+fby2).5/Fb <=1.0   3.3.1-2

For fa/Fa =< 0.15

fa/Fa + (fbx2+fby2).5/Fb <= 1.0   3.3.1-3

Axial Tension + Bending

In accordance with API clause 3.3.2 members are proportioned to satisfy equation 3.3.1-2

fa/0.6Fy + (fbx2+fby2).5/Fb <=1.0   3.3.1-2

5.6.2 Combined Bending in Non-Tubulars

The maximum permissible combined loads are checked against the requirements of AISC clause H1.

Axial Compression and Bending

For fa/Fa > 0.15

$$\frac{fa}{Fa} + \frac{Cmxfbx}{(1-fa/F'ex)Fbx} + \frac{Cmyfby}{(1-fa/F'ey)Fby} =< 1.0 \qquad \text{H1-1}$$

fa/0.6Fy + fbx/Fbx + fby/Fby =< 1.0   H1-2

For fa/Fa =< 0.15

fa/Fa + fbx/Fbx + fby/Fby <= 1.0   H1-3

Axial Tension + Bending

fa/0.6Fy + fbx/Fbx + fby/Fby =< 1.0   H2-1

5.7 Hydrostatic Loading

If the depth of water above the model origin is entered than the code checker will check the hydrostatic loading in accordance with the requirements of API Clauses 3.2.5, 3.3.3 and 3.3.4.

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

The maximum effective length (**EffLx** or **EffLy**) is used as the cylinder length.

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

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

-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

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.

AISC 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

C10 Depth of water above the mode origin (Hydro collapse check - see Section 4)

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

Using Groups to Sort Output

G1 Group SET to read

G2 Group Limit/Restriction

G3 Groups Only switch

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

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

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

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

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

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

-o-