# FS2000

## *Tubular Joint 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 TubeJoint is an interactive program module that interfaces with FS2000 to provide tubular joint design checks in accordance with the following design code specifications.

**CHS Chord & Braces**

American Petroleum Institute (API) "RP2A Recommended Practice for Planning, Designing and Constructing Fixed Offshore Structures, 22st Edition (2014)

BS EN ISO 19902:2007 Petroleum and natural gas industries - Fixed steel offshore structures

American Petroleum Institute (API) "RP2A Recommended Practice for Planning, Designing and Constructing Fixed Offshore Structures, LFRD (1989)

Department of Energy: Offshore Installations "Guidance on design, construction and certification" 4th Edition 1990.

CIDECT Design Guides /BS 5950:Part 1(4)/Eurocode 3: Annex K(3).

The American Institute of Steel Construction (AISC) "The AISC Specification for Structural Steel Buildings" AISC 360-16 Edition.

**RHS Chords & RHS/CHS Braces**

CIDECT Design Guides /BS 5950:Part 1(4)/Eurocode 3: Annex K(3)

Data required for the module is obtained from FS2000's model data and analysis results. The element properties are referenced by the same property and material codes as in FS2000. Each joint is examined to establish the chord elements and the connected braces. Non tubular elements and joints are ignored when using the API and DEn codes.

The operation of the program is by interactive menus or batch command line operation. The Module can produce Summary output or Stress Output. Summary Output gives only the Unity Ratios whereas Stress output additionally gives the actual and allowable for the component actions. In Interactive Mode, more detailed information on the joint design parameters can be obtained.

The Unity Ratio plot routines in FS2000 can be used to plot the utilisation ratios obtained from the output. These can also be used to show those joints that have been checked.

**Type of Joints Checked**

The following types of joint will be checked (Note: API and DEn only check CHS/CHS joints).

| Chord | Brace | Axial | In-plane | Out-plane | T/Y | X | K/N |
|---|---|---|---|---|---|---|---|
| CHS | CHS | Y | Y | Y | Y | Y | Y |
| CHS | I/H (1)* | Y | Y | Y | Y | Y | N |
| CHS | RHS (2) | Y | Y | Y | Y | Y | N |
| | | | | | | | |
| RHS | CHS | Y | Y | Y | Y | Y | Y |
| RHS | RHS(3) | Y | Y | Y | Y | Y | Y |

AISC is based on EC3/CEDEC but does omit the following check that will be made in accordance with EC3: In-plane RHS to RHS moments and CHS to I/H connections.

For these ( ) brace joints to be recognised the following must be satisfied. If they are not satisfied a warning will be given.

- The major axis (Iz in FS2000) ie the web axis must align with the axial axis of the chord.

- A principle axis (y or z) of the brace must align with the axial axis of the chord and a principle axis of the chord must align with the axial axis of the brace.
- A principle axis (y or z) of the brace must align with the axial axis of the chord and a principle axis of the chord must align with the axial axis of the brace.

**Warnings**

If the program module is operated in Interactive Warnings will be given for all joints that are not be checked. The warnings list is not given in the Summary Mode or Stress Model output.

The UR plot function has an option to show all recognised joints.

-o-

## 2.0 Creating Models in FS2000

This section describes the few requirements necessary when creating models that are going to be code checked and methods used for basic joint type Recognition/Classification. Recognition/Classification is described in Section 2.2.

-o-

## 2.1 Units

TubeJoint reads the inputs and result files of models created and analysed by FS000. 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 E values and material yield etc

or US-Unit i.e.

Force in Lbs (Pounds)

Length in ins (Inches)

Stress in psi (Pounds/Inches**2) E values and material yield etc

The section properties are identified by the program and the length units are converted to mm and the stress units to N/mm2.

-o-

## 2.2 Joint Type Recognition/Classification

All joints between tubular member are examined to identify chord and brace members.

In CHS joints the member with the largest diameter is taken to be the chord member.

In RHS or mixed joints the RHS chord is identified as the member with the largest perimeter/width. For equal size members the brace with a continuation members is selected. For equal external sizes without a continuation member, the member with the largest csa is selected. All other members are regarded as braces. Joints of tapered members will be based on the smaller end.

**Chord Recognition** - The following methods maybe used to identify chords

- In CHS X joints in which chord and brace members are the same diameter it may be necessary to fractionally increase a member OD to ensure that the member is identified as the chord. In RHS joints the stress points data may be similarly increased.
- The chord can be explicitly declared (see Section 4).
- If members are identical and no chord continuation member exists, the member with the highest label will be selected as the Chord

It is not necessary for the chord to be continuous across the joint for the member to be identified as a chord. However, the chord continuation member will be recognised as such providing the member alignment is within about 5 deg. The chord continuation member need not be the same size as the chord. Chord stresses are averaged across the joint. Chord members require to be rotationally aligned for correct bending stress averaging.

Individual brace joint classification can be according to the loading pattern in the joint in the same plane as the respective brace or the geometric configuration of the brace. For braces that carry part of their load under different classifications their classification is based on the interpolation of the respective portions of

each in total. In the program this is termed force joint classification.

By default all braces will be automatically classified but the user may re-classify any individual brace. See Sect 4.1 for brace classification methods.

Joints are classified as K, Y, X or XW for the purpose of joint capacity.

They are geometrically classified as TY, K1, K2, KT or XT joints. If a brace has a member on the opposite side of the joint it is first classified as an X joint. If the brace has an adjacent brace on the same side it is re-classified as a K1 joint. If there are more than two braces on the same side the joint is re-classified as a KT joint, with the central brace (largest angle) being classified as a K2 brace and the outer braces as KT braces.

-o-

## 3.0 Operation of the CodeChecker

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

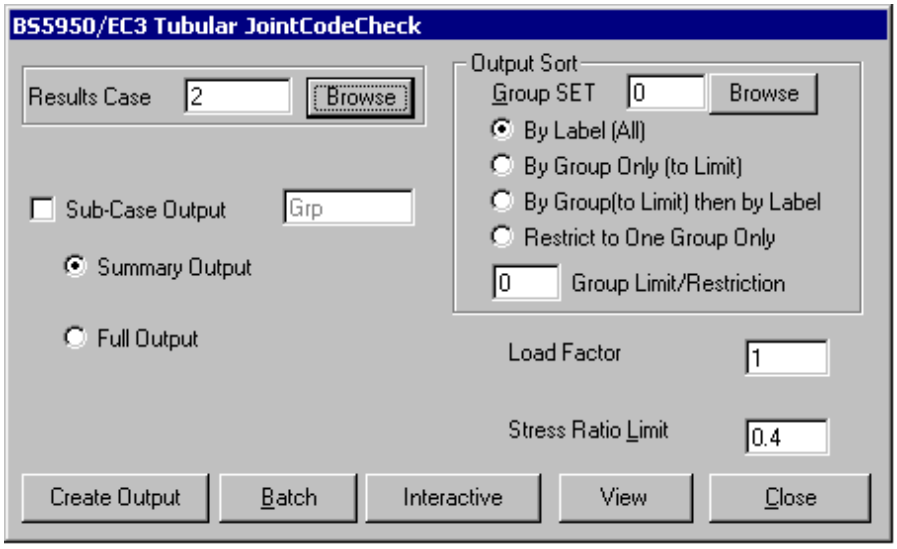
The CodeChecker can be run from FS2000 in direct mode using the Summary Check or Stress Check options. In this mode the output results are created without any further interaction with the user.

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

-o-

## 3.1 Operation from FS2000

When this commands is selected the following type of 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.

**Summary Check** option selects the summary mode which shows only unity ratio output

**Stress Check** option produces the stress mode which produces output the shows the actual and allowable loading.

**Allowable Stress Factor** is used to enter a factor to increase the allowable stress ie the 1/3 increase normally associated with extreme environmental loading would be input as 1.333. This is only used in the API code checks.

**Load Factor** only used by the CIDECT code check. If unfactored load are being used (Allowable stress design) then a value of 1.5 to 1.6 would be typical for normal live loading. A value greater than 1 wil reduce the load capacity. If factored loads are being used this value should be unity or a value to represent the material partial load factor e.g. Load Factor = $\gamma$m/1.1 would be used for design to EC3. Note that when load factors are used it is the allowable load that is effected, not the applied load.

The **Operating** and **Extreme** options are used in the DEn code check

**Stress Ratio Limit** is used to restrict the output to those braces with maximum unity ratios that 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 **Interactive** button activates Interactive operation of the CodeChecker

The **Batch** button converts the set options to command line switches and appends the PUNCH 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.

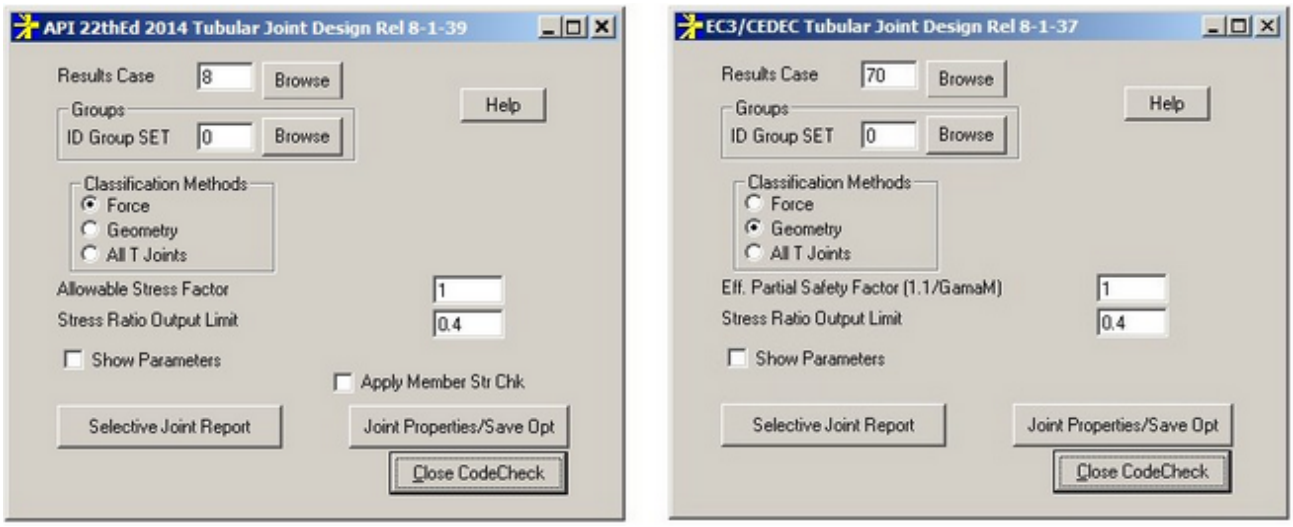
**Important Note** Stress ratio data created by this module will be limited to the data processed. If restricted process options is used then any Stress Ratio plots or Stress Ratio sorts which use the same results cases will be limited to the processed data. The plot or listed output will indicate if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




The **Classification Methods** box is used to select the method of global joint classification. The current option is saved when the **Save Modified Data** button is clicked in the **Joint Properties** option form. (JAF command in the <model>.ECJ file)

> **Force** joint classification is evaluated according to the loading pattern in the joint in the same plane as the respective brace. The net classification is based on interpolation on the respective portions of each in total. E.g. a K joints could act as a 20% K :80% T joint in terms of load balance on the joint. Note that the API Chord Load factor is always based on the geometrical joint type.
>
> **Geometry** joint classification uses the geometrical configuration of the joint as the bases for the capacity check.
>
> **All T Joints** will classify all braces as T joints.
>
> **All K Joints** will classify all braces as K joints
>
> **All X Joints** will classify all braces as X joints

**Allowable Stress Factor** is used to enter a factor to increase the allowable stress ie the 1/3 increase normally associated with extreme environmental loading would be input as 1.333. This is only used in the API code checks.

**Eff. Partial Safety Factor (1.1/GamaM)** is used by the EC3/CEDEC code check. It is a method of defining the material partial resistance factor. If a different material load factor from 1.1 default is to used then this has to be changed. If partial load material factor of 1.05 was required then this factor would be set to 1.1/1.05 = 1.0048. If non-factored load are being used (Allowable Stress Design) then a value of 1.5 would be typical for normal loading.

**Design Format** is used by the AISC/CEDEC check. 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

The **Selective Joint Report** button is used to select individual or sets of joints for selective codecheck. When the option is selected a Selection form will become visible. This selection form is used to define the joints(s) (Node) to be checked. The output from this mode is listed on the screen in a list viewer. It may also be printed directly from this viewer.

The **ID Group SET** is used to select elements by group attribute.

If an individual joint is specified and the **Show Parameters** option is active then a more detailed output showing some of the relevant factors used in the equations appropriate to the type of joint being checked will be shown. If multiple joints are selected only a summary output will be given.

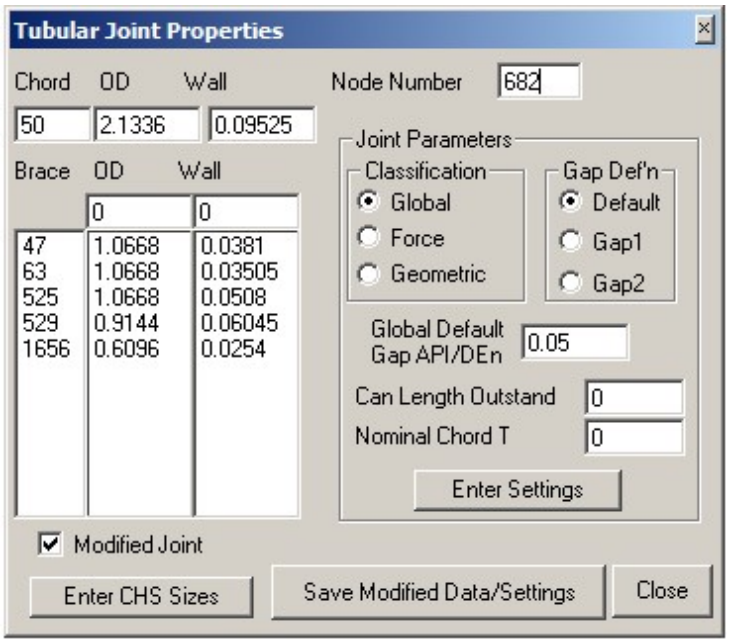
The **Apply Member Str Chk** will include the API Minimum Capacity check. Use the JSR command to (edit the **<model>.ECJ** file) change the the strength ratio from the default value of 0.5 (50%). Note that these parameters (JST & JSR) will be added to the ECJ file only when when the Save Modified Data/Settings button is clicked (Tubular Joint Properties form).

The **Joint Properties/Save Opt** button is used to view and define joint properties and also save the settings on this form. When it is clicked the Joint Properties form will become visible. The **Save Modified Data/Settings** button on that form is used to save the settings on both forms.

-o-

## 3.3 Modification of Joint Properties

When the Joint Properties button is pressed the Tubular Joint Properties form shown below will appear. This form is used to re-define data relating to the code checking of specific joints.




The **Node Number** box is used to enter the node label number of the joint whose properties are to be inspected/modified. The data relating to the joint will appear in the other boxes when a joint is entered.

The **Chord** boxes show the element label of the joint chord and it's OD and Wall thickness. To change the OD and Wall thickness simply enter the desired values and click the **Enter CHS Sizes** button.

The **Brace** list shows the element labels of the connecting braces. To change brace OD and Wall thickness select the brace in the list and then edit the values in the upper edit boxes. Click the **Enter CHS Sizes** button the enter the data.

The **Modified Joint** check box indicates if a joint has been modified ie the properties at the joint are not those of the property code of the member. If this value is un-checked and the **Enter CHS Sizes** button is clicked the chord data will revert to the property code sizes for a that joint.

The **Joint Parameters** frame is used to enter data relating to both CHS and RHS joints. The Global Default Gap is used to define the default gap for K joints used in the API & DEn for code checks. (this is the JGG command in **<model> .ECJ** file)

The **Classification** options are described in Section 4.1

The **Gap Def'n** options are described in Section 4.2

The **Global Default Gap API/Den** defines the global joint gap to be used.

The following two parameter are used to evaluate the chord capacity for thickened chord cans. The strength reduction is not applied to the K proportion of the joint classification.

The **Can Length Outstand** (Lo) is used to evaluate Lc, the effective chord can length. Lc = 2*Lo + d/sinθ. If Lo=0, the default then Lo= the greater of D/4 or 0.3. If Lo is defined as a -ve value the Can Reduction factor will also be applied to the moment capacity.The value of Lo used is listed in the EGJ file.

The **Nominal Chord T** (NCT) has to be specified if the can geometry is defined with the model geometry. This must be zero if the chord can is defined as a Design Parameter. The value of NCT used is listed in the EGJ file.

The **Enter Setting** button must be clicked first before the Save Modified Data button if the Joint Parameters options are to be changed to those shown in the form.

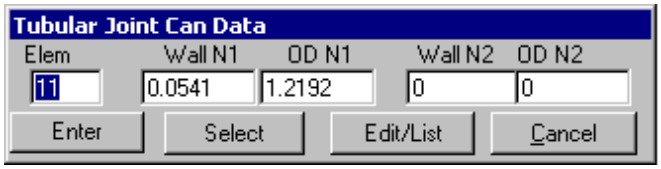
The **Save Modified Data/Setting** button is used to save the all the modified parameters in this form. The modified parameters are saved in an ASCII file **<model>.ECI**. This button also saves setting in the Joint Parameters frame to the **<model>.ECJ** file.

-o-

## 3.4 Joint Design Parameters - Graphical Definition

The joint geometric parameters ( wall thickness and diameters) 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 assign data to elements.

The following Tubular Joint Can Data Input form becomes visible when the Define command is selected from the TASK/Design/CanDesign menu.




Wall N1 Member wall thickness at Node1 of the element.

OD N1 Member outside diameter at Node1 of the element

Wall N2 Member wall thickness at Node2 of the element

OD N2 Member outside diameter at Node2 of the element

Zero values are ignored and infer that the property code values will be used in be used in cases where no previous definition exists. Exiting values will not be changed if a zero value is entered in a box.

If the Select button is clicked the elements will be selected by the current selection method.

This Edit/List button enables existing element data to be listed, edited or copied. When the button is pressed the mouse is used to select the element is to be listed. The data can now be modified or copied as required using label reference or the current selection method.

***Visual Selection***

If an element is listed with the Element Query button the First Node of the will be circled. This identifies to which boxes (N1 or N2) the data should be entered

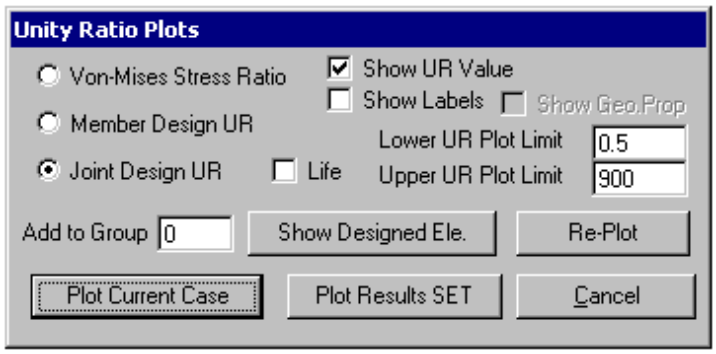
It is not possible to define classification in the graphics environment but it is possible to interrogate the classification once the code checker has run.

-o-

## 3.5 Plotting Design Utilisations

The Unity Ratio plot selection form in FS2000 GUI (Output/Resulsts TASK) can be used to show the design utiliations of all checked braces.

This started from the UR(Unity Ratio) command in the Plots menu or the UR button in the Results Too lBar.


The **Plot Current Case** button is used plot the URs of the active Results Case. If Sub-Cases exist a list will appear. The appropriate sub-case is that selected by mouse click. Note that the this buttons only becomes active following the opening of a Results Case

The **Plot Results SET** button is used to plot the URs from a number of Results Cases. This option reads the results in the Results SET and plots for each element only the highest UR in the SET. Along side the UR value is the results case that produced this maximum. If Sub-Cases exist they will be included in the plot and be identified by their case number.

The **UR Plot Limits** are used set a upper and lower visibility limit. All elements withURs within these limits will be plotted red.

The **Add to Group** box is used to group all the elements within the UR Limits to the specified Group. This useful when wishing to restrict output to highly loaded members in other output modules e.g. text output.

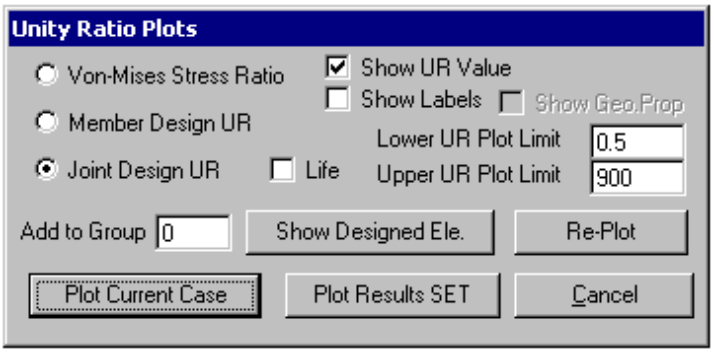
The **Re-Plot** button is used to re-plot the results following changes in the UR plot parameters e.g. limits.

The **Show Designed Ele** button is used to show those elements that have been designed checked. Elements that have been design checked are highlighted red.

-o-

## 3.5 Showing Check Joints

The Unity Ratio plot selection form can be used to show which brace elements at a node have been code checked.




The **Show Designed El** button is used to show all brace elements that have been code checked. If an element is shown in red then it has been recognised by the code checker and checked accordingly. When selected, a result case will require to be selected.

The facility is also useful for indicating which elements at a node are the chord elements since chord elements will not be red unless they become at brace elements at a different connecting node.

-o-

# 4.0 Joint Classification

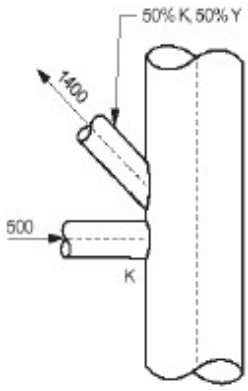
## 4.1 Brace Classification Code

The brace classification code is used to define which method is to be employed for individual joint brace classification that are identified by their node label. The following classification codes are used.

D Use Global Classifications. Geometric configuration, Force-classification or T joint specification (this defined in the Interactive Operation form - see 3.2)

F Use Force Classification

G Use Geometric Classification




This is the JFC command in **<model>.ECJ** file,

**JFC**, *Joint Node, Classification Code (D, F or G)*

See Appendix A for the file formats.

The Force Classifications evaluated by the program will be shown in the output. It is recommended that the classifications be checked on critical or complex joints.

```
*** Actual/Allowable Joint Loading ***
Node     Chord      Brace    Angle    Type    Axial       In-Plane    Out-Plane
         O.D.        O.D.             (Gap)   Allow       Allow       Allow     CombUR
          t            t                      UR1         UR2         UR3        MaxUR

2       1           3          45.00    KT    1.414E+06   2.000E-04   1.000E-04
         1.500     0.800             0.900    7.036E+06   3.326E+06   1.603E+06   0.201
         0.040     0.012    50Y 50K 0X            0.201       0.000       0.000   0.201

                   4          90.00    K2   -5.000E+05   2.000E-04   1.000E-04
                  0.800             0.050    6.876E+06   2.352E+06   1.134E+06   0.073
                  0.012    0Y 100K 0X            0.073       0.000       0.000   0.073
```

-o-

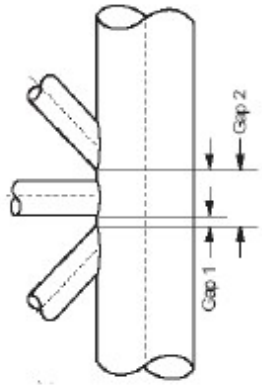
## 4.2 Gap Codes & Individual Gap Definition

The gap codes define how brace gaps are to be defined. The following code are used

0 Use Global default values. (this defined in the Joint Properties Form - see 3.3)

1 Use GAP1 evaluated by the program ( see 4.3.2)

2 Use GAP2 evaluated by the program ( see 4.3.2)




This is the JGD command in .ECJ file.

**JGD**, *Joint Node, Gap Code (0,1 or 2)*

**Specific Gap Definition**

Gap definition may also be specified by definition within the JBC command in the **<model>.UJClass** (Joint classification file). Definition here will override the above codes. See Appendix A for a summary of data files and format.

-o-

## 4.3 Data Formats for User Joint Classification

### 4.3.1 Joint Classification File (**<model.>.UJClass**)

This file is created by the user (in a text editor) to define chord elements, individual brace gaps and related brace classifications. If this file exists it will be interpreted by the program.

All data fields must be entered

**Chord Identification**

**JCH**, *Joint_Node_Number, Chord_El_Label, Chord_Continuation_El_Label*

Enter 0 if there is no chord continuation member

**Brace Classification**

**JBC**, *Joint_Node_Number, Brace_Element_Number , Gap, Classification, Multi_Planer, 0*

Valid classifications are : N None, will use methods defined elswhere (see 4.1).

T

K

X without diaphragms

XW with diaphragms

*Multi_Planer* is a de-rating factor applied to multi-planer joints

### 4.3.2 Calculated Joint Gaps Output File (**<model>.EJG**)

This file shows the brace gaps evaluated by the program. The file is not used by the program. Its purpose to indicate how the joint is interpreted by the program. It is useful for identifying gaps and overlaps. Overlapped joints are identified by negative values. More often these value will be negative.

***For API and DEn Codes***

For K1 and K2 braces the default gap (GAP) is based on the Global Default Gap.

For KT braces the default gap (GAP) is based on 2xGAP plus the projected length of the central brace.

***CIDECT Codes***

For RHS K1 and K2 braces the default gap (GAP) is based on the larger of 0.5(bo-bi) and 2ti.

For CHS K1 and K2 braces the default gap (GAP) is based on ti + tj

For KT braces the default gap (GAP) is based on 2xGAP plus the projected length of the central brace.

***EJG File Data Format***

The file format is:

Node Number, Element Number, Geometric Classification, GAP1, GAP2, GAP, e, e/h, Lo, NCT

Geometric Classifications are those established by the program.

GAP1 is the gap in a K joint or the gap between the outer braces in a KT joint.

GAP2 is the gap between the central brace and the outer braces in a KT joint assuming that the

central brace is centrally located between the outer braces.

GAP is the gap used by the program

e is the K joint eccentricity based on GAP

e/h is the K joint eccentricity ratio based on GAP. h is the chord depth or diameter

Lo is the can outstand length used by the program.

-o-

# APPENDIX A - Data Files

### Files associated with the Code Checker

***Definition***

**<modelname>.ECI** Modified Tubular Joints - Individual user defined chord and brace size definition for CHS joints. Note: When saved in Design TASK the data will be ordered by element label. When saved in the code checker it will be ordered by node label.

Commands

**CN**, *Element, Node, Wall, OD*

***User Defined Classification***

**<modelname>.UJClass** Individual user defined brace/gap definitions.

Commands

**JCH,***Node, Chord, ChordCont*

**JBC**,*Node, Brace, Gap, Class, Multi-Planer, 0*

**<modelname>.ECJ** Joint classification codes, gap directives and other global settings.

Commands

| | |
|---|---|
| **JFC**, *Joint Node, Classification Code* | *(D, F or G)* |
| **JGD**, *Joint Node, Gap Code* | *(0,1 or 2)* |
| **JGG**, *Global Gap* | |
| **JAF**, *Global Classification Method* | *(0 - Force, 1 - Geometry, 2 - All T/Y)* |
| **JST,** *API Minimum Capacity Check* | *(0 or 1, 1 activates) Clause 7.2.3* |
| **JSR,** *Brace Strength Ratio API Strength Check* | *(typically 0.5 or 1.0)* |
| **CEL**, *Joint Node, Lo* | *Can Outstand Length (Lo is used to evaluate Lc API Clause 7.3.5)* |
| **NCT**, *Joint Node, T* | *Nominal Chord Thickness (Used to evaluate Pa API Clause 7.3.5)* |

***Results***

| | |
|---|---|
| <modelname>.J"m" | Tubular Joint Output text file-Full |
| <modelname>.K"m" | Tubular Joint Output text file-Summary |
| <modelname>.6"m" | Tubular Joint Stress Ratio file |
| <modelname>.EJG | Secondary information file on classifications and gaps (See 4.3.2 for EJG file description) |

-o-

## APPENDIX B - Batch Operation

This appendix defines the command line options for the codechecker. For further details of command line operation refer FS200 Help or manual.

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

C1 Results FileC2 1- Full Report 2 - Summary Report

C3 Unity check ratio limit for output

C4 Allowable stress increase factor (Load Factor for CIDECT)

C5 Code to be used. Enter API or DOE

C6 Operating condition E for extreme, O for Operational

E or O is only used in DEn code check.

C7 Group SET to read

C8 Group Limit/Restriction

C9 Groups Only switch

C10 Subcase name

For C7-C9 (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-