RULES FOR MODELLING STRUCTURES

CONTENTS

Section 3 b

RULES FOR MODELLING STRUCTURES

Introduction

The idealisation of the real structure into an acceptable geometrical grid, set of elements, sizes, constraints, materials, loading etc. has but one purpose, to enable the REAL structure loads, stresses and deflections to be calculated. The idealisation should thus represent all the load paths and realistic loading distributions set up.

At the stage of formulating the size and scope of the analysis, decisions will be made which force the idealisation of the structure along fairly recognisable paths. For example in the modelling of a fuselage a coarse mesh with skin panels bounded by major frames would imply the use of shear and rod elements for the skins.

Unless the idealisation is carried out keeping in mind the full requirements, pitfalls are numerous. Overkill of the mesh size may create a dinausaur with no future; over simplification can leave out load paths which may be critical. The use of pre-processors such as Patran gives the engineer the ability to generate complex structures with ease, but this does not necessarily mean that they will be more accurate.

Intrinsically the solution of the stiffness matrix, application of loading and constraints, the generation of displacements, and the back substitution to obtain loads and stresses is an accurate MATHEMATICAL process. This does not imply that the results from any model are correct. Accuracy should not be confused with correctness. The correctness of the result is in the hands of the user.

Overall accuracy, no doubt, remains linked to the source and quality of the information available. It is pointless setting up a complex analysis based on preliminary scheme drawings, approximate loading and uncertain materials. The question "How accurate is the available data?" is rarely asked, and if qualified would generally lead to suprising tollerances on the results.

The process of idealisation requires geometry to be defined and element sizes to be calculated which satisfy area or stiffness requirements, and only rarely both. Thus in most instances the model is a compromise, and the subsequent interpretation to the real structure is treated differently.

The stress engineer should not use analysis results blindly. Modelling, loading and constraint errors should be eliminated in the checking process, but may still be present. Thus be on guard for unexpected results.

The user should only launch into the idealisation process having first determined for himself a clear method of approach and a good understanding of the total requirements.

SELECTING THE GEOMETRY

There are two stages in determining the model geometry, locating the source, and its application to the model. Both of these stages have several options.

Source of geometry :- The source can play a vital part in the scope of an analysis, not only in the accuracy but in the elapsed time to definition. A wing analysis, for example, would require a greatly extended time scale if the surface geometry was not available. If key diagrams for a structure are not available a complex analysis should not be undertaken since by implication the results would be approximate.

a) Defined explicitely .. drawings giving all the required geometry. This may be input directly into Nastran or Patran.

b) Scale drawings .. these may be interpreted several ways, ..hand measurement, using a digitiser, or if available on Anvil by conversion into a mesh via Patran.

c) Key diagrams .. giving the intersect points of major items, ribs/spars etc. The mesh may be created using Anvil and converted to a Nastran set of points or passed to Patran for further refinement.

d) Surfaces .. available on Anvil or Catia. These are used to form up the basic model using intersects as patch boundaries which are then put to Patran for further refinement, or to generate profile geometry by projecting a previously defined 2-D mesh of points onto the surface.

e) Solid geometry .. Catia items. The process of conversion to an analysis grid is currently being studied and remains a specialist activity.

Application to model :-

The geometry may be used directly in some cases, but in other cases should be modified to represent the best interests of accuracy or expediancy to the user.

Some comments are given below.

a) Flying surfaces, wings etc .. are relatively thin structures and thus in order to keep the skin membrane loading and stiffness correct, the mid skin thickness geometry should be used. There is a proceedure within the Optimisation routines which will carry out this task automatically.

b) Fuselage shells .. a decision should be made on the geometry profile which should be consistant, ie. at the outer CPL, the inner skin point, or the mid skin point. Rules for calculating frame flange areas etc can thus be fixed. Because of the relative size errors are small whichever approach.

c) Fuselage frames .. sloping frames should be converted to local axes so that the in plane stiffness is not lost due to small kinks. PPS offers an automatic conversion of co-ordinate systems in option 6.

d) Fuselage floors, shear webs .. ensure the geometry is planar, convert to local axes if nesessary.

e) Ring frames .. modelled as shear webs and rods, geometry at skin points and internal points (observe 3 above).
.. modelled as bar elements, geometry as 2 above and use offset vectors to section N/A.

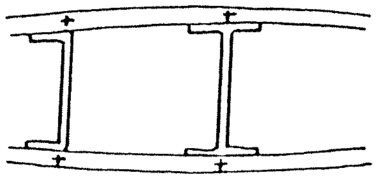
f) Longerons/spars .. ensure that these follow the correct (straight) lines. Geometry errors which create kinks both reduce the effective stiffness and impose unreal loads on the support points.

g) Offset joints .. beware of setting up joints involving offsets from the main load paths unless the offset has been modelled correctly and the local structure is capable of reacting the kink loads. It is better to put the joint coincident with a skin/frame/longeron junction. The joint loads can be re-calculated and offsets taken into account in the subsequent detailed stressing work.

h) Hinge lines .. the geometry of the hinge points must be set up co-linear in order to avoid 'locking up'. Use a coordinate system with the hinge line axis nearest to the basic x, y, or z direction. See the section on the modelling of Hinges.

GEOMETRY - APPLICATION TO MODEL

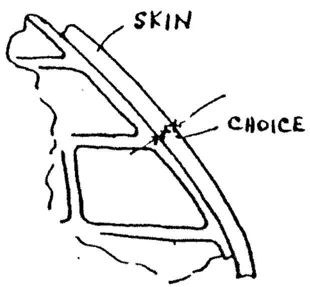
WINGS ETC:-



GEOMETRY AT MID SKIN DEPTH - PRESERVES SKIN LOADING AND SECTION STIFFNESS

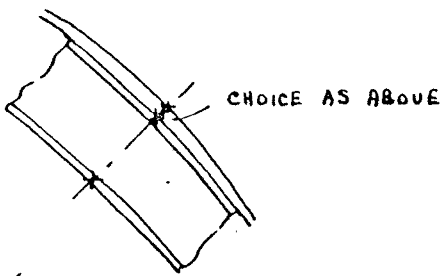
FUSELAGE SHELLS :-

USE CONSISTANT GEOMETRY



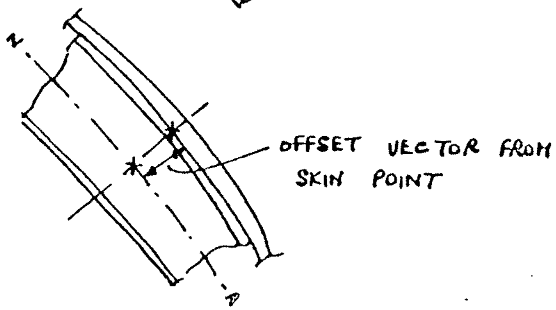
RING FRAMES :-

SHEAR PANEL / BOOM IDEALISATION

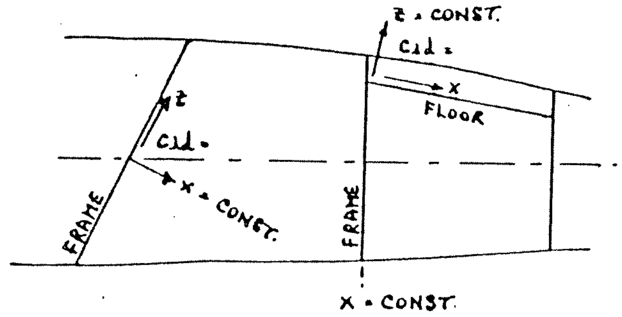


OR :-

BAR IDEALISATION



SLOPING FRAMES, FLOORS, SHEAR WEBS :-

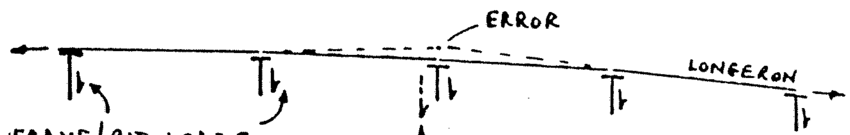


KINKS CA
LEAD TO
MAJOR LOS
OF IN-PLA
STIFFNESS

AVOID KINKS IN GEOMETRY - PUT INTO LOCAL AXES MAKING ONE CO-ORDINATE CONSTANT.

LONGERONS, SPARS ETC :-

AVOID KINKS IN GEOMETRY

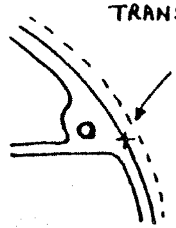


FRAME/RIB LOADS DUE TO CURVATURE ETC.

UNREAL LOAD DUE TO GEOMETRY ERROR PLUS LOSS OF STIFFNESS IN LONGERON.

OFFSET JOINTS :-

TRANSPORT TYPE JOINTS



USE GRID AT NEAREST SKIN POINT.

PREVENT LONGERON OR SKIN KINK LOAD - DETAIL STRESS WILL COVER THIS.

IDEALISATION - RULES FOR MODELLING STRUCTURES

GENERAL

The idealisation of the real structure into the Nastran model should, as previously stated, only be undertaken when the full purpose and scope of the analysis has been determined.

What should be clear is :-

The size of the grid
The position of all major load paths
The interface points to other componants or between frames and shell etc.
The loading and support requirements. Additional points may have to be created.

Nastran is a displacement method program - it solves for displacements at grid points. Thus the general aim in the idealisation process is to model the CORRECT STIFFNESS. This normally requires the specification of the correct thickness and area of the items, but in many instances these are modified due to a compromise in the selection of the geometry.

If a single element represents several items with different areas or materials, the calculation of the idealised size will be linked to the selected idealised material stiffness, and the subsequent interpretation of the results will have to use the same relationships to obtain the real structure loads.

Nastran element forces and stresses balance the applied loading, however STRESSES are only accurate for the real structure in the instances where the GEOMETRY is at the centroid of the item and the element SIZE is correct. Thus in most instances the element FORCES are used in the subsequent interpretation the real structure. The user must ensure the element and real structure forces are in balance when correcting for geometry and size idealisation methods (or changes to the structure).

The following page illustrates some general idealisation and interpretation methods.

INTERPRETATION OF RESULTS

IDEALISATION INVOLVES A NUMBER OF FORMAL OPERATIONS TO DETERMINE THE STRUCTURAL EQUIVALENCE –

THIS MUST BE **REVERSED** TO PROVIDE CORRECT RESULTS ON THE **REAL** STRUCTURE.

GEOMETRY
ELEMENT TYPE
SIZE (AREA, THICKNESS)
BALANCE

} **ALL RELATED**

DETAIL STRESS CALCULATIONS

NASTRAN IS A DISPLACEMENT METHOD PROGRAM – SOLVES FOR DISPLACEMENTS AT GRID POINTS

ELEMENT FORCES AND STRESSES BALANCE THE APPLIED LOADING

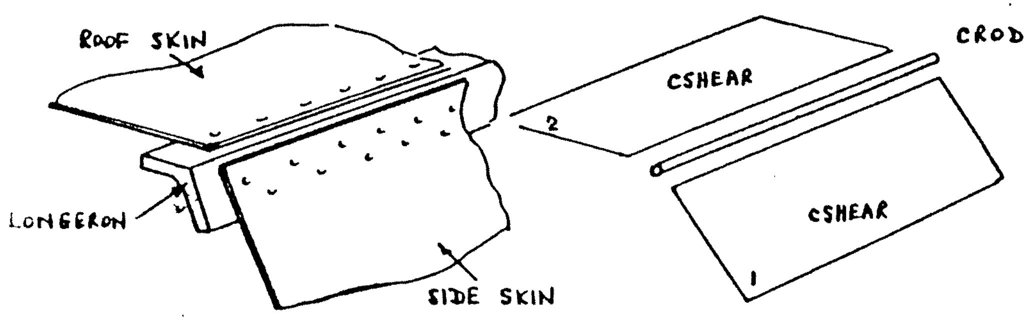
HOWEVER NASTRAN STRESSES ARE ONLY ACCURATE FOR THE REAL STRUCTURE IN A FEW INSTANCES

a) WHERE THE GEOMETRY IS AT THE CENTROID OF THE ITEM

AND b) WHERE THE ELEMENT SIZE IS ACCURATE

USER MUST ENSURE ELEMENT AND REAL STRUCTURE FORCES ARE IN BALANCE WHEN CORRECTING FOR GEOMETRY AND SIZE IDEALISATION METHODS, OR CHANGES TO THE STRUCTURE.

DIRECT AREA RELATIONSHIPS

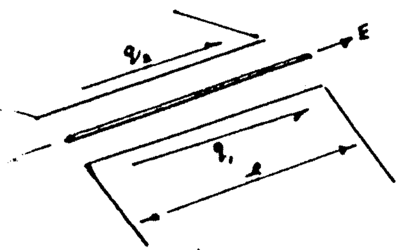


ROD AREA = AREA OF LONGERON + EFFECTIVE AREAS OF SKINS

IF MATERIALS ARE DIFFERENT SCALE AREAS BY RATIO OF $\frac{\text{ACTUAL E}}{\text{CROD E}}$ BEFORE ADDING

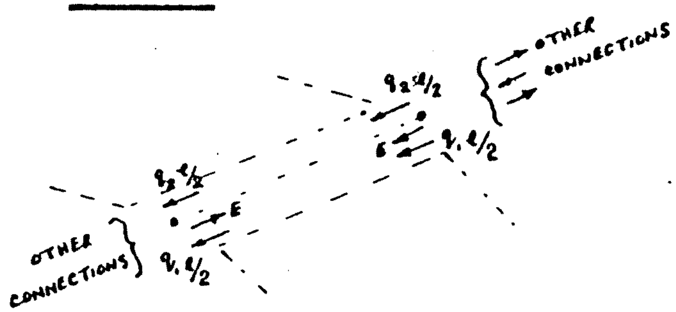
NASTRAN OUTPUT

FORCES



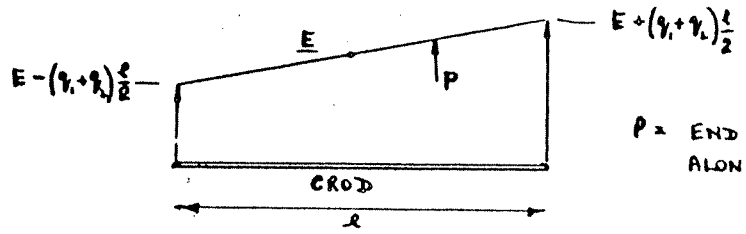
...ND LOAD IN CROD
...EL EDGE SHEAR FLOWS

GPFORCES



SINCE GPFORCES ARE IN GLOBAL SYSTEM, THEY MUST BE RESOLVED INTO CROD AXIS

LOADS IN CROD



P = END LOAD AT POSITI... ALONG ROD

END LOAD KNOWN AT 3 POINTS

LOAD IN LONGERON $= \frac{P \cdot A_L}{A_{TOT}}$

END LOAD IN PANELS $= \frac{P \cdot A_P}{A_{TOT}}$

CONSTANT STRESS IN ITEMS $\sigma = \frac{P}{A_{TOT}}$

A_L, A_P, AND A_{TOT} ARE AREAS USED TO EVALUATE PROD ARE...

IF MATERIALS ARE DIFFERENT

STRESS IS GIVEN BY $\sigma = \frac{P \cdot A_P}{A_{TOT}} \cdot \frac{E_P}{E_{CROD}}$

PANEL LOADS

SHEAR LOADS OBTAINED FROM FORCE OUTPUT ON EDGES
END LOADS OBTAINED FROM EFFECTIVE AREA AS ABOVE
STRESSES OBTAINED FROM E.L. AND SHEAR FLOWS AND RATIO OF IDEALISED TO ACTUAL THICKNES

ie $\sigma = \frac{q}{t_I} \cdot \frac{t_I}{t_{ACT}}$

FASTENER LOADS

RIVET OR BOLT SHEAR LOADS $= \frac{q\,\ell}{N}$ OR $\frac{q}{\text{PITCH}}$

WHERE q = EDGE SHEAR, ℓ = PANEL LENGTH,

N = NUMBER OF FASTENERS, PITCH = FASTENER PITCH

Conveniently the various A/C componants can be covered seperately and the recomended modelling techniques are shown below. The element sizing is linked to the selection of the geometry and the types of element chosen, thus in the examples that follow the sizing method is shown explicitely.

INTERPRETATION of the results is linked to the method of idealisation and thus guidelines are included where convenient.

Fuselage Frames

These can be in various forms, ring frames, machined part/full frames or pressure bulkheads.

Light Ring frames :- Two methods of idealisation are possible.

a) Shear web and boom. The frame is represented as a shear web and boom using CSHEAR and CROD elements and sized to combine the effective web and flange areas. The geometry should be set at the average depth between flanges and flange efficiency factors should be used in determining the CROD areas.

Stressing method : use the bending moment calculated from the boom end loads and shear values to carry out detailed stressing round the frame.

It is assumed that the fuselage skin effective area is modelled as a separate CROD and thus the frame loading can be readily found. If the outer CROD includes the effective skin, some load sharing is nesessary before the outer flange load can be determined and thus the frame bending moment.

b) Bar representation. CBAR elements are specified using the outer geometry and offset vectors to the N/A of the section. The frame bending moment and shear values are output directly. Make sure that the bars are co-planar (eg X=const) and if not supply all properties A,I1,I2,J etc. Remember the default for shear flexibility is ZERO (infinitely stiff). Note that additional D.O.F. are required for this method.

Stressing method : Some interpolation is required since the skin shears are applied at the grid points and thus steps appear in the BM at the grid points - use the average. The moments at the centre of the elements will be correct.

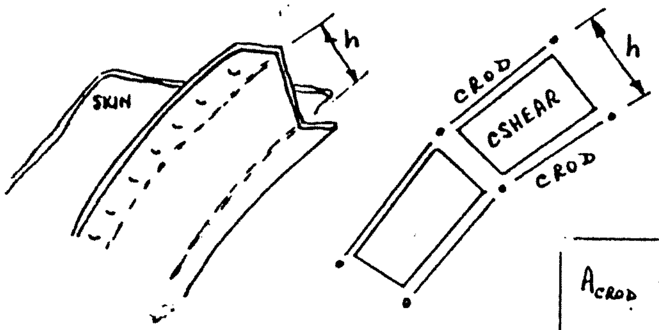
Heavy Ring Frames :-

The idealisation is calculated to maintain the correct moment of inertia and N/A position, the geometry being dictated by the fuselage skin and internal skin if present. The stresses in the real structure can then be calculated using E.B.T. and the analysis stress levels.

A method is shown below :-

RING FRAMES - LIGHT SECTIONS

a) SHEAR WEB & BOOM



GEOMETRY SET AT AVERAGE HEIGHT h.

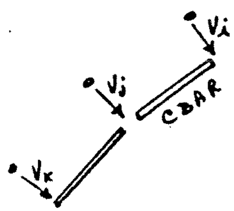
CROD AREAS NEED NO CORRECTION.
Efficiency Curve Calculated Using ESDU Datasheet.

$$A_{CROD} = \text{FLANGE AREA} \times \text{EFFICIENCY} + \frac{\text{WEB AREA}}{6}$$

STRESSING METHOD :- USE ANALYSIS GEOMETRY AND CROD FORCES TO PRODUCE BENDING MOMENT DISTRIBUTION ROUND FRAME; CSHEAR LOADS FOR SHEARS

SKIN EFFECTIVE AREA IS A SEPARATE CROD ELEMENT,
IF NOT --- LOAD IN OUTER FLANGE ELEMENT MUST BE SHARED TO DETERMINE FRAME LOADING.

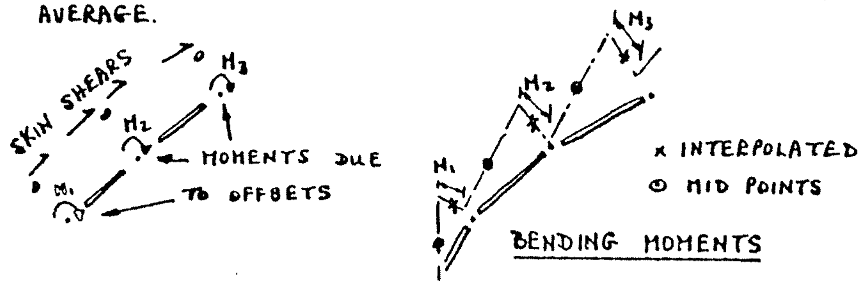
b) ALTERNATIVE IDEALISATION .. USING BAR ELEMENTS



REQUIRES :- CALCULATION OF OFFSET VECTORS, SECTION PROPERTIES, ADDITIONAL ROTATIONAL D.O.F. IN MODEL.

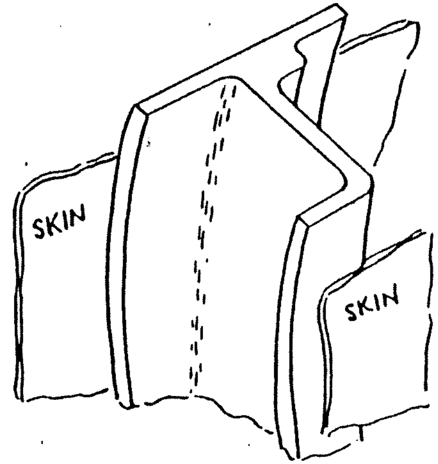
NOTE :- USE ~~CBARϕ~~ CARD TO REQUEST OUTPUT AT POINTS ALONG BAR.

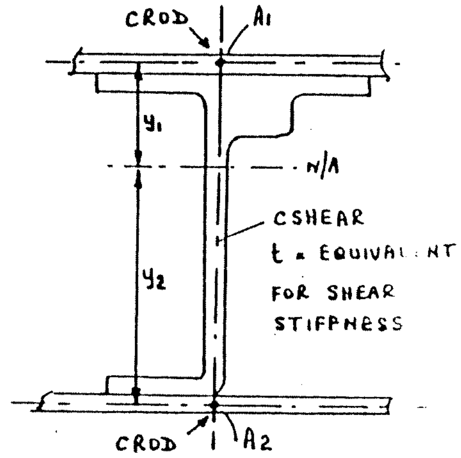
STRESSING METHOD :- NASTRAN OUTPUT IS END LOAD, MOMENT AND SHEAR ROUND FRAME. MOMENT AT ELEMENT ENDS IS INPUT FROM SKIN SHEARS AND THUS IS STEPPED - USE AVERAGE.



BENDING MOMENTS

HEAVY RING FRAMES





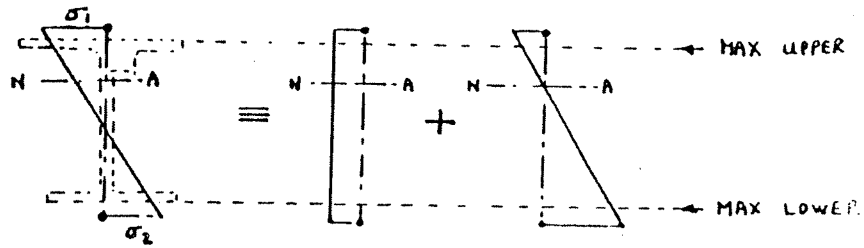
CALCULATION OF CROD AREAS A_1 AND A_2

1. FIND FLANGE EFFICIENCY FACTORS USING E.S.D.U. 71004
2. CALCULATE SECTION I AND POSITION OF N.A. - REDUCE FLANGE AREAS BY EFFICIENCY FACTORS AND FLANGE WIDTH FOR CALCULATION OF LOCAL I's.

$$\left.\begin{aligned} A_1 y_1^2 + A_2 y_2^2 &= I \\ A_1 y_1 &= A_2 y_2 \end{aligned}\right\} \text{HENCE } A_1 \text{ AND } A_2$$

STRESSING METHOD :-

USING NASTRAN STRESSES IN CRODS, σ_1 AND σ_2 CALCULATE STRESS USING LINEAR INTERPOLATION OR E.B.T.



Machined frames :- The idealisation should follow the stiffener pattern as close as possible, using CRODS for the stiffeners and CSHEAR's for thin (non end load effective) panels and CQUAD4's for thick end load carrying panels. Provided the areas and thicknesses are accurate no corrections to the real structure are nesessary.

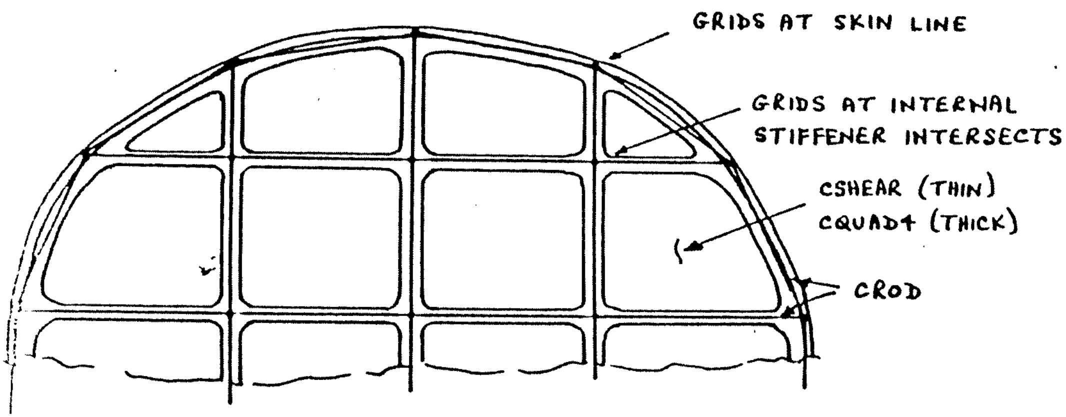
CSHEAR's will give the panel edge shears directly whilst QUAD4 elements quote the centre of panel values (in element axes). Using the GPFB for a QUAD4 panel will enable the edge shears and loads to be calculated - remember to resolve the loads into the axis of the edge concerned - GPFB's are produced in GLOBAL axes.

Pressure Bulkheads :- For an initial simple idealisation of a fuselage model the pressure bulkheads are usually modelled for in plane loading only, the pressure loading effects being taken into account in the detailed stressing. Thus all stiffeners have the correct area but are not subject to bending loading. This enables the model to be kept simple and does not involve any additional rotational D.O.F.

However, for more detailed work the bending effects should be modelled and pressure loading supplied as part of the loading case definition. The stiffeners should then be modelled using CBAR's.

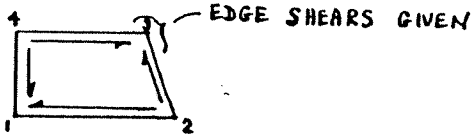
The CBAR's should be used with offsets to their correct N/A position - in which case the bending moments will be correct at the centre of the elements and need averaging at the ends. Idealisations without offsets would require re-calculation of the bar loads to take into account the offset shear loads.

MACHINED FRAMES



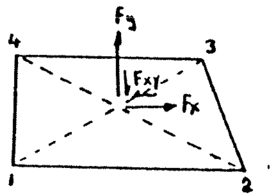
PROVIDED ROD AREAS AND PANEL THICKNESSES ARE CORRECT NO CORRECTIONS ARE NECESSARY.

CSHEAR FORCES IN PANELS :-

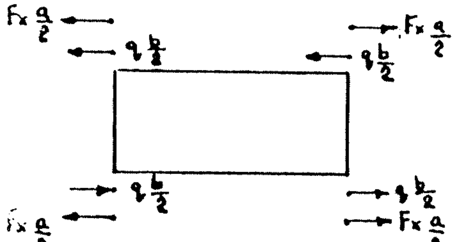


CQUAD4 FORCES IN PANELS :-

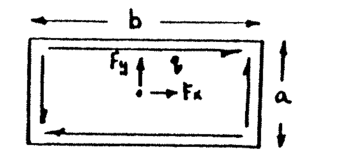
PANEL LOADS GIVEN IN CENTRE OF PANEL IN ELEMENT AXES



USE G.P. FORCE OUTPUT RESOLVED INTO THE PANEL EDGE DIRECTIONS TO OBTAIN EDGE SHEARS AND END LOADS.
FOR RECTANGULAR PANELS :-



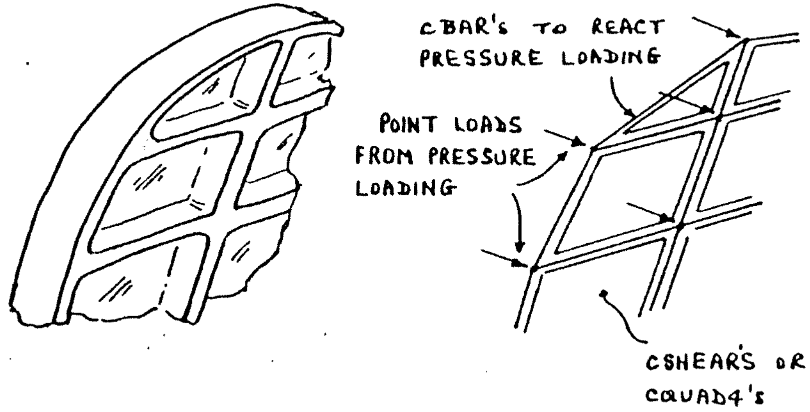
X - DIRECTION LOADS



AVERAGE SHEAR STRESS σ_{xy}
SHEAR FLOW $q = \frac{\sigma_{xy}}{t}$

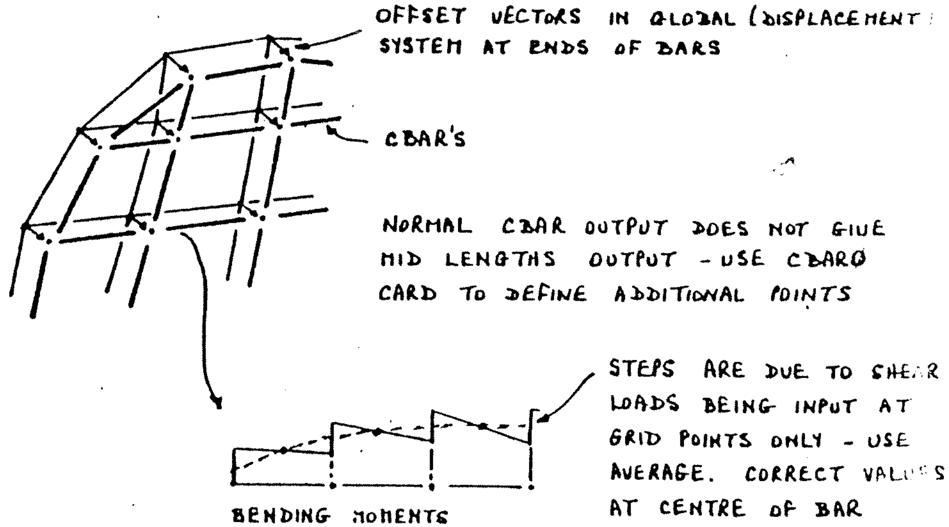
PRESSURE BULKHEADS

(See the Existing Analysis for the Exa..
ie EFA or EAP Pressure Bulkheads)



CBAR END LOAD, MOMENTS AND SHEARS ENABLE STIFFENER STRESSES TO BE CALCULATED, MAKING CORRECTIONS FOR BAR OFFSETS.

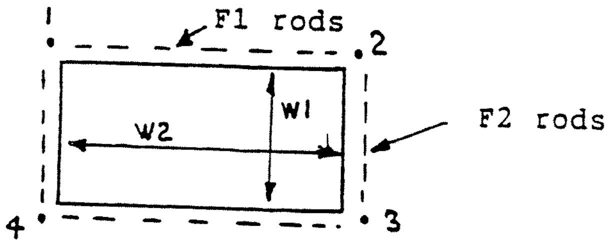
ALTERNATIVELY - USE CBAR'S WITH OFFSET VECTORS



Note on the use of CSHEAR elements

The CSHEAR element normally carries shear loads only and thus should be surrounded by ROD elements to carry end loads in the equivalent areas. If this extensional stiffness is not present the panel will be singular and would probably lead to a FATAL error or excessive displacements.

A facility to give the shear element extensional stiffness in either direction exists using the F1 and/or F2 factors. (See PSHEAR description).



F1 = factor for areas on sides 1,2 and 3,4

F2 = factor for areas on sides 1,4 and 2,3

If F1 or F2 are set to 1.0 the equivalent rods would be set to areas of 1/2.t.W1 or 1/2.t.W2 ... ie. fully effective for end loa

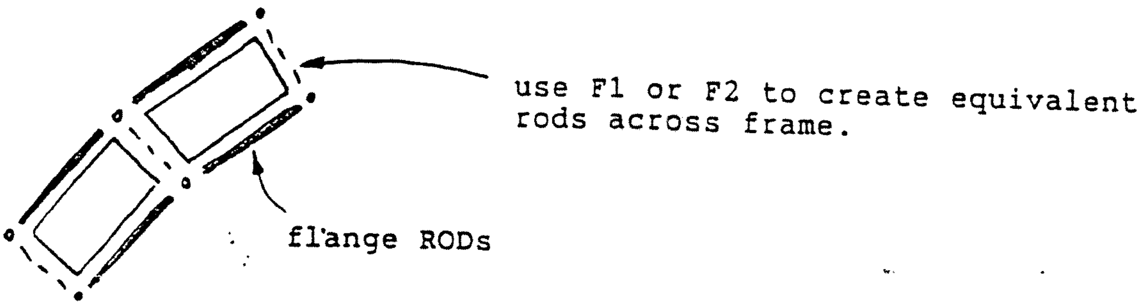
If the panel is less than fully effective, two methods of approach are possible.

a) F < 1.01, the areas are set to 1/2.F1.t.W1 or 1/2.F2.t.W2 .. ... a function of the panel WIDTH.

b) F > 1.01, the areas are set to $1/2.F1.t^2$ or $1/2.F2.t^2$ .. ... a function of the panel THICKNESS.

USE

These factors can be used to remove the singularity for ring frames across the frame depth.



AND

In calculating flange areas the web thickness componant Aw/6 can be included using the F factors by setting the required F factor to 1/3.

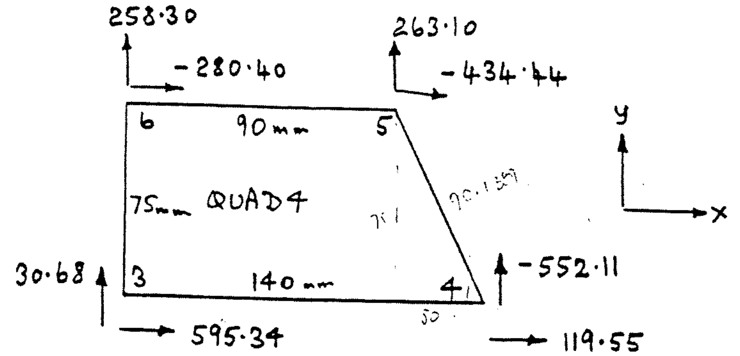
WARNING ... Using F1 or F2 = 1.0 would lead to an over stiff bending section.

DIRECT LOADS IN PANELS .. The only method of determining the end loads carried in the SHEAR panel when an F factor is used is to look at the corner forces F-FROM-4 F-FROM-2 etc. Without F factors the corner forces are of equal magnitide and ammount to the shear loading on the edges. With F factors the corner forces will be different and the shear contribution should be removed to determine the end loads in the panel. For rectangular panels orthogonal to the global axes the corner forces are identical to the GPFB values ....see next page.

UNDERSTANDING NON-RECTANGULAR PANELS

For rectangular panels the GPFB output can be easily used to calculate the shear and end load components of the panel loading, and these will be identical to the panel FORCE output for both CSHEAR and CQUAD4 panels...see pages 6 and 10.

For non-rectangular panels the GPFB output can be used to calculate the edge loads provided the output is transformed into the edge directions and components from adjacent edges removed. Consider the example below.

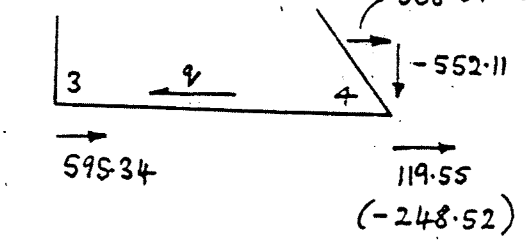


GPFB LOADS AT CORNERS

$\tan\theta = 75/50 = \frac{552 \cdot 1}{x}$

$x = 368 \cdot 07$

EDGE 3-4



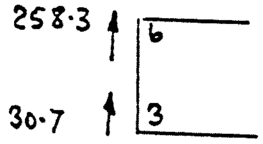
368·07 = COMPONENT OF EDGE 5-4 LOAD

(-248·52) ← SUBTRACT 5-4 COMPONENT

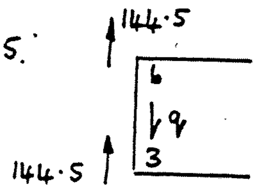
SHEAR CONTRIBUTION = ½ SUM

→ 173·4 → 173·4 $q = \frac{173 \cdot 4 \times 2}{140} = \underline{2 \cdot 48 \text{ N/mm}}$

EDGE 6-3

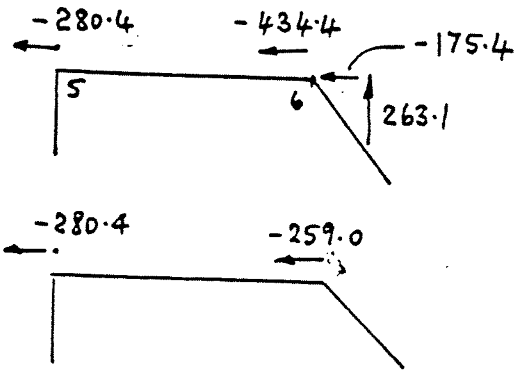


½ SUM = 144·5

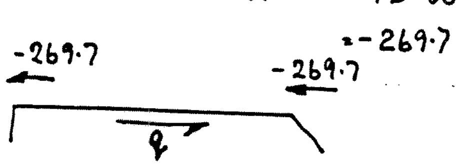


$q = \frac{144 \cdot 5 \times 2}{75} = \underline{\underline{3 \cdot 85 \text{ N/mm}}}$

EDGE 5-6

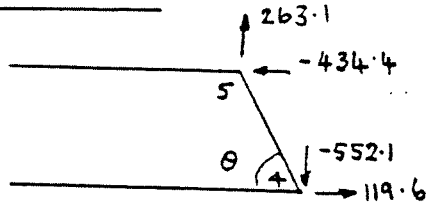


SHEAR CONTRIBUTION = ½ SUM

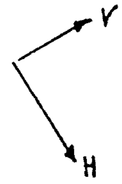


$q = \frac{269 \cdot 7 \times 2}{90} = \underline{\underline{5 \cdot 99 \text{ N/mm}}}$

EDGE 4-5

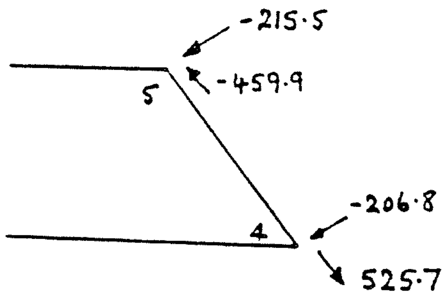


LET THE DIRECTIONS BE

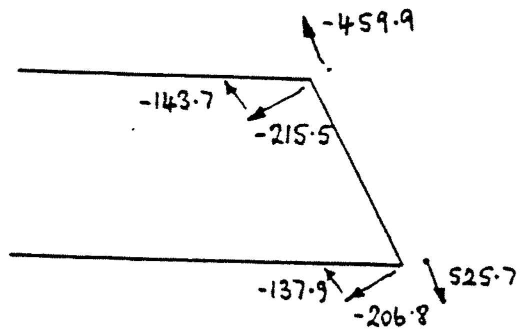


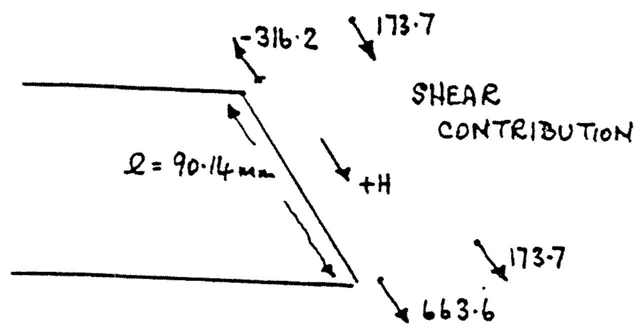
$$\theta = \tan^{-1}\frac{75}{50} = 56 \cdot 3099^\circ$$

RESOLVING GPFB LOADS INTO H & V AXES GIVES



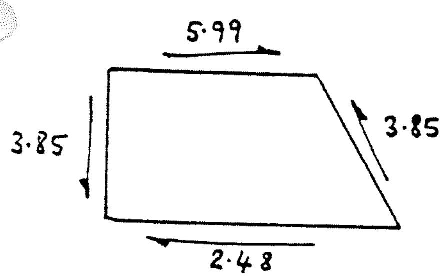
REMOVING ADJACENT EDGE COMPONENTS





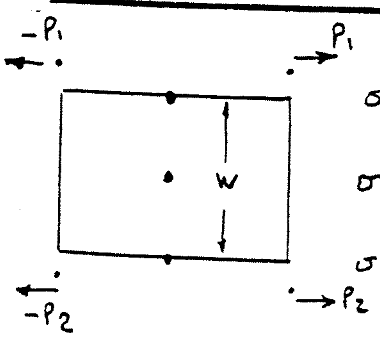
$$q = \frac{173 \cdot 7 \times 2}{90 \cdot 14} = \underline{\underline{3 \cdot 85 \text{ N/mm}}}$$

FINAL PANEL EDGE SHEARS



PANEL SHEAR FROM QUAD4 OUTPUT = 5·37 N/mm.

CALCULATION OF PANEL END LOADS



$$\sigma = \frac{P_1}{\tfrac{1}{2}wt}$$

$$\sigma = \frac{P_1 + P_2}{wt}$$

$$\sigma = \frac{P_2}{\tfrac{1}{2}wt}$$

PANEL END LOADS CAN BE CALCULATED USING THE ABOVE EDGE LOADS ... ½ DIFFERENCE ON EACH GRID. THIS END LOAD IS THEN EFFECTIVE OVER HALF THE PANEL WIDTH AND THUS MAXIMUM AND MINIMUM STRESS LEVELS CAN BE CALCULATED AT THE MID SIDE POINTS

EXTRACTING END LOAD AND SHEAR TERMS FROM GPFB.

1) RECTANGULAR PANELS

- IF GEOMETRY IS NOT ORTHOGONAL TO GLOBAL AXES RESOLVE INTO EDGE DIRECTIONS

$\longrightarrow \frac{F_1+F_2}{2}$ $\qquad$ $\longrightarrow \frac{F_1+F_2}{2}$ SHEAR CONTRIBUTION IS ½ SUM

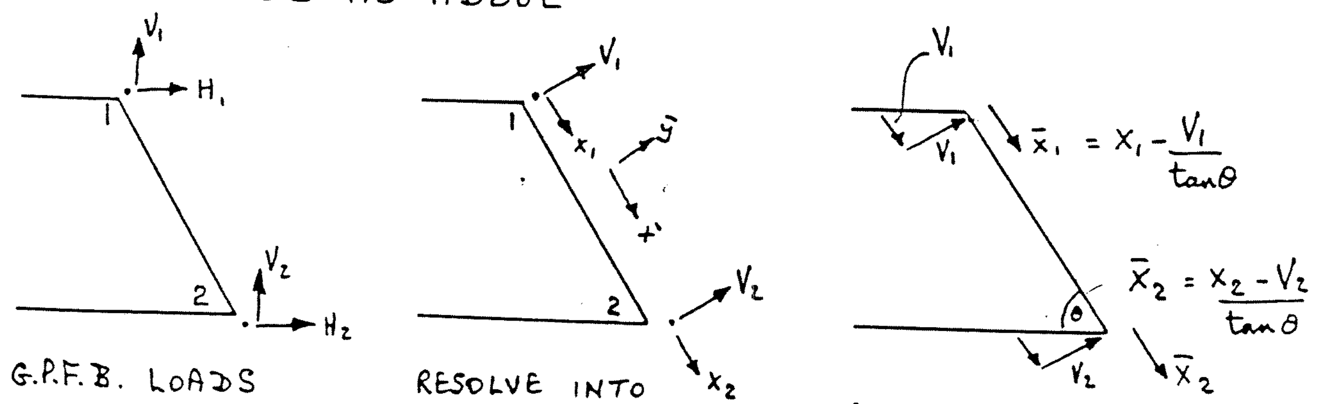
$\longrightarrow \frac{F_1-F_2}{2}$ $\qquad$ $\longleftarrow \frac{F_1-F_2}{2}$ E.L. CONTRIBUTION IS ½ DIFFERENCE

PANEL EDGE SHEAR $q = \frac{F_1+F_2}{\ell}$ WHERE ℓ IS 1-2 LENGTH

MID EDGE E.L. STRESS $\sigma = \frac{F_1-F_2}{\frac{1}{2}W\,t}$ WHERE W IS PANEL WIDTH NORMAL TO 1-2. t IS THICKNESS

2) NON-RECTANGULAR PANELS

- FOR EACH EDGE RESOLVE LOADS INTO EDGE DIRECTIONS
- REMOVE ANY COMPONANTS FROM ADJACENT EDGES
- PROCEED AS ABOVE



G.P.F.B. LOADS $\qquad$ RESOLVE INTO EDGE DIRECTIONS $\qquad$ REMOVE CONTRIBUTIONS FROM ADJACENT EDGES

- USING $\bar{X}_1$ AND $\bar{X}_2$ PROCEED AS IN 1)

NOTE :- USE OF PARAM NOELOF 1 WILL OUTPUT $\bar{X}_1$ AND $\bar{X}_2$ LOADS DIRECTLY BUT OF OPPOSITE SIGN.

Use of PARAM NOELOF to obtain panel edge loads

GPFB output is presented at the corner grids of panels with forces given in the GLOBAL directions for the grids concerned. Only if the panel edges are orthogonal to the global directions can the GPFB terms be used directly to calculate the edge shears and direct loads. If the panel is skewed a complex process such as on the previous pages is necessary to deduce the edge loads.

There exists within Nastran a facility to output edge loads directly, and from these the shears and direct loads are easily calculated using the 1/2 sum and difference method.

The output is requested by using a PARAM card in the BULK DATA

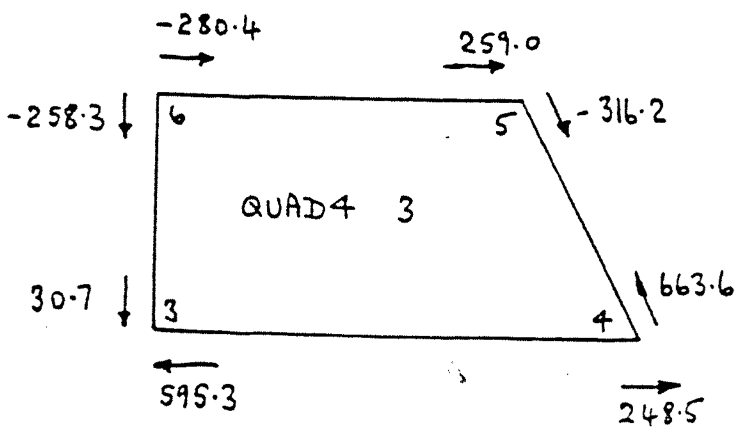
PARAM NOELOF 1

The panel/element loads in the edge axes are output in a similar manner to the GPFB. Loads are positive in the direction of the reference point to the load point. Vector wise these are opposite to the GPFB loads. This is because the EIF forces are Element Internal Forces and thus balance the GPFB loads.

A typical example of output for the previous example is shown below.

E L E M E N T   I N T E R N A L   F O R C E S   A N D   M O M E N T S
(INTERNAL ACTIONS FROM REFERENCE-POINTS TO LOAD-POINT)

LOAD POINT-ID	ELEMENT -ID	ELEMENT TYPE	REFERENCE POINT-1	FORCE-1	MOMENT-1	REFERENCE POINT-2	FORCE-2	MOMENT
3	2	QUAD4	2	2.093722E+02	0.0	6	-1.004949E+02	0.0
3	3	QUAD4	6	3.067943E+01	0.0	4	5.953369E+02	0.0
3	8	ROD	2	8.041086E+02	0.0	0	0.0	0.0
3	9	ROD	4	4.181440E+02	0.0	0	0.0	0.0
3	12	ROD	6	6.981548E+01	0.0	0	0.0	0.0
4	3	QUAD4	3	2.485225E+02	0.0	5	-6.635554E+02	0.0
4	9	ROD	3	4.181440E+02	0.0	0	0.0	0.0
4	10	ROD	5	-5.382942E+02	0.0	0	0.0	0.0
5	3	QUAD4	4	-3.162063E+02	0.0	6	2.590393E+02	0.0
5	6	QUAD4	6	-1.136974E+02	0.0	12	-3.883735E+02	0.0
5	10	ROD	4	-5.382942E+02	0.0	0	0.0	0.0
5	11	ROD	6	-1.453421E+02	0.0	0	0.0	0.0
5	23	ROD	12	-4.661270E+02	0.0	0	0.0	0.0
6	2	QUAD4	3	2.988660E+02	0.0	7	4.045027E+02	0.0
6	3	QUAD4	5	-2.804495E+02	0.0	3	-2.583323E+02	0.0
6	5	QUAD4	7	-5.285955E+02	0.0	11	-3.344448E+02	0.0
6	6	QUAD4	11	3.771284E+02	0.0	5	2.654961E+02	0.0
6	11	ROD	5	-1.453421E+02	0.0	0	0.0	0.0
6	12	ROD	3	6.981548E+01	0.0	0	0.0	0.0
6	13	ROD	7	-3.620258E+01	0.0	0	0.0	0.0
6	21	ROD	11	6.766563E+01	0.0	0	0.0	0.0



EDGE LOADS FOR PANEL 3 FROM ABOVE OUTPUT.
THESE ARE IDENTICAL TO THOSE CALCULATED ON PREVIOUS PAGE.

N B

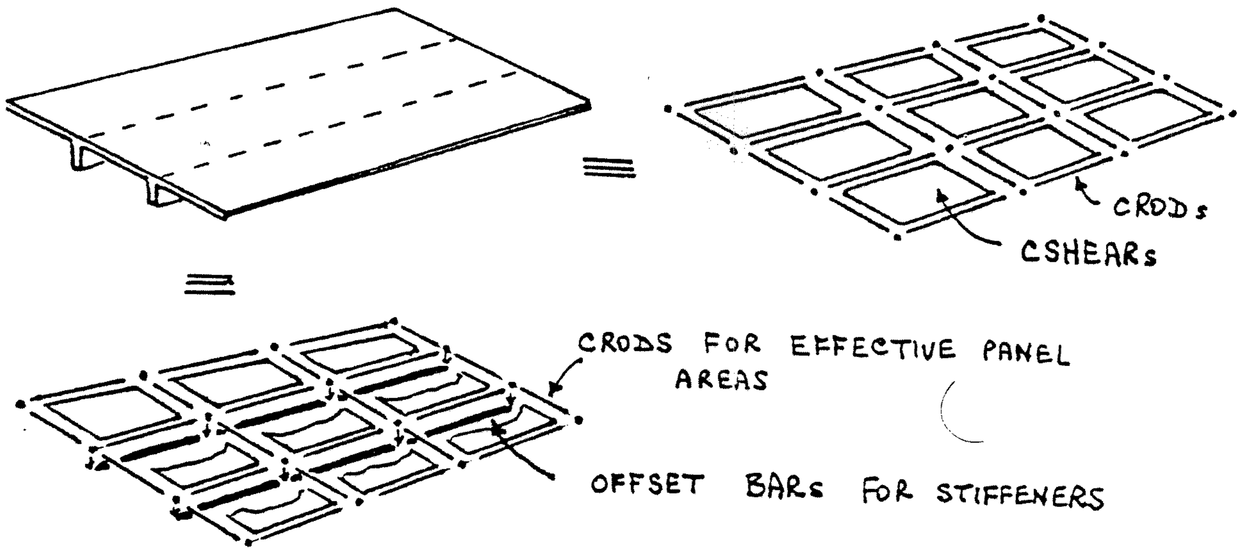
OUTPUT EQUAL AND OPPOSITE TO GRID POINT FORCE BALANCE OUTPUT

FLOORS

Usually these can be regarded as being fully effective in end load and shear, and may be required to carry pressure loading. The normal idealisation is by using ROD and SHEAR elements, the ROD elements representing any stiffener areas plus the effective panel areas.

If the panel has any significant stiffener offsets and/or is subject to pressure loading, BARs should be used with offset vectors, leaving the RODS only for the panel end load paths.

For initial project work the pressure effects can be added in the detail stressing, the model reduced to in plane effects only and thus no rotational D.O.F. are needed.



The panel effective end load areas can be set up automatically by setting F1 and F2 = 1.0 on the PSHEAR card, in which case each effective rod area will be 1/2 the panel width x thickness.

GEOMETRY .. make sure the floors are planar, put into a local axis system if nesessary. A slight out of plane kink in any of the grids will destroy the load carrying capability.

N.B IF BUCKLING PERMITTED THEN IDEALISATION MUST TAKE ACCOUNT OF THIS BY EITHER REDUCING 'E' OR 'A' TO [illegible] THICKNESS. NOTE THEN THAT ANALYSIS IS ONLY CORRECT FOR BUCKLED STATE

WINGS - RIBS, SPARS, SKINS

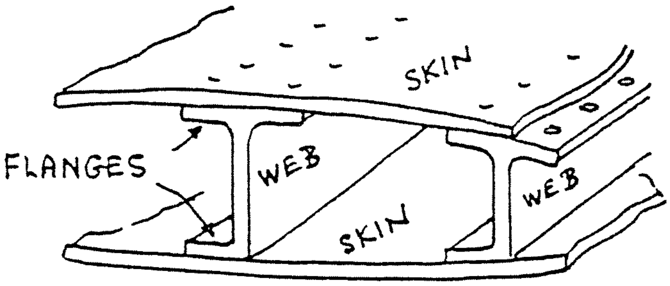
Wings, fins etc. are thin structures domonated by bending loading for the skin design and shear loading for the internal spar/rib construction. Thus the idealisation should aim at providing the correct stiffness for these items.

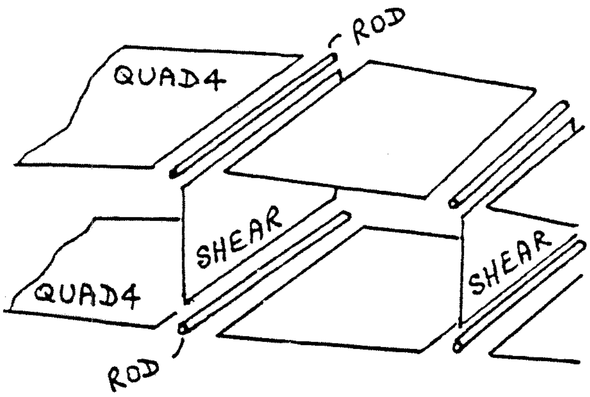
The geometry should be set up at the mid depth of the skins, thus the loading and stiffness will be correct. The spar/rib idealisation should aim at obtaining the correct moment of inertia - in a similar way to that for heavy ring frames.

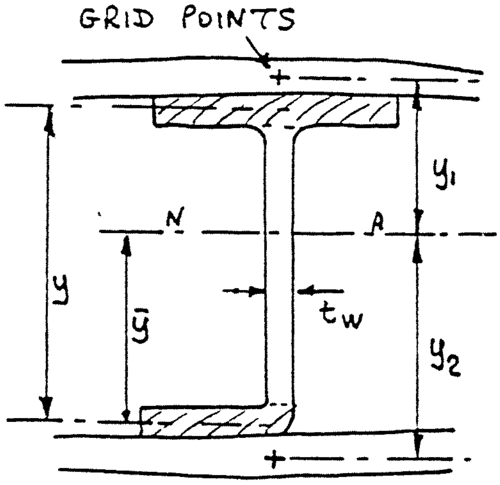
The skins should be idealised as QUAD4 elements, the spar/rib webs as SHEARs and the flanges as RODs. Since the CSHEAR element only has ROD elements along the skin edges, some stiffness must be given in the through thickness direction, thus use the appropriate F factor on the PSHEAR card. Alternatively use RRODs, linking only the through thickness D.O.F. between top and bottom skin grid points. If one of these methods is not used excessive displacements will be generated since the webs will be near singular.

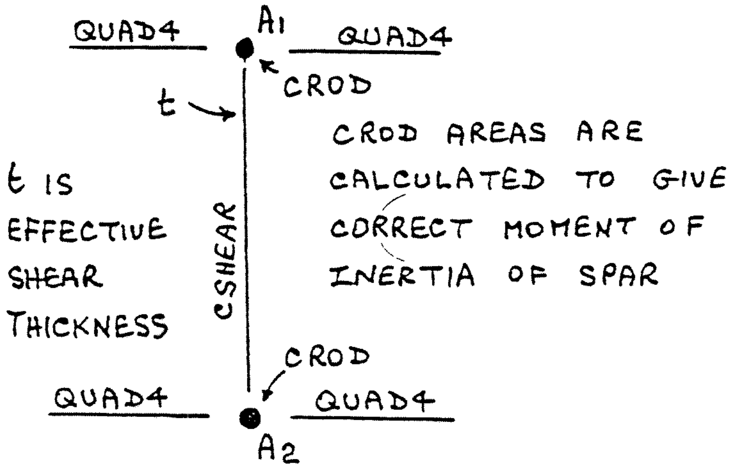
WINGS ETC. - RIBS AND SPARS

IDEALISATION SHOULD REPRESENT <u>SECTION</u> <u>STIFFNESS</u>









GEOMETRY IS SET UP AT SKIN MID DEPTH - QUAD4 LOADS ARE ACCURATE IF SIZES ARE CORRECT

IDEALISED ROD AREAS CALCULATED FROM :-

$A_1 y_1^2 + A_2 y_2^2 = I$ ... SPAR/RIB MOMENT OF INERTIA

$A_1 y_1 = A_2 y_2$ ... MAINTAINING N.A. POSITION

HENCE A_1 AND A_2

<u>STRESS</u> IN UPPER FLANGE IS GIVEN BY $\sigma_u = \frac{\sigma_{A1}.(y-\bar{y})}{y_1}$

AND IN LOWER FLANGE BY $\sigma_L = \sigma_{A2} . \frac{\bar{y}}{y_2}$

<u>NOTE</u> :- USE PSHEAR F1 OR F2 FACTOR TO GIVE THROUGH-THICKNESS SPAR/RIB END LOAD STIFFNESS.

FUSELAGE SKINS

The idealisation of fuselage skins is probably the most difficult to model correctly in that the modelling is dependant on several factors :-

a) the size of the mesh
b) the frame support
c) the skin thickness
d) the design condition, buckled/unbuckled etc

However it is possible to divide the methods into two groups, coarse mesh and fine mesh.

Coarse Mesh Fuselages :-

In the coarse mesh idealisation the skin between the modelled frames is represented by single elements. It is not possible to represent any skin bending (pressure loading will load the frame grids only) or hoop tension effects - these must be taken into account in the detail stressing work. The skins are thus represented by SHEAR elements and the effective end load areas by ROD elements.

Longetudinally the skin will be fully effective and thus the F factor on the PSHEAR card may be used to provide this, in which case longetudinal CRODs will represent any longeron or stringer areas. Alternatively if the F factor is not used the CROD areas should represent the sum of the effective end load and longeron/stringer areas.

The CRODs along the panel edges bounded by the frames will be sized depending upon the effective area of the skins in the circumfrential direction. ESDU 71004 gives formulae for the effective areas for single and multiple frame attachment lines. The CRODs will be independant of any frame flange areas, which should have been modelled separately.

For thick skins the effective areas may approach 100% (intake ducts etc.), and in this case the CSHEARs may be replaced by CQUAD4s, ie fully end load effective. In this case CRODs would be required only for the longerons etc.

The design condition will dictate the material stiffness used, ie. a reduced E for a buckled structure.

WARNING ... if QUAD4s are used for skin elements globally the structure could/will be significantly over-stiff, which would lead to incorrect load paths and deflections.

Fine Mesh Fuselages :-

In a fine mesh situation the skins between frames are modelled in sufficient detail to enable skin bending and hoop tension effects to be predicted. This forces the model to have rotational D.O.F. and this increases the running time considerably. The mesh should be sufficiently fine (minimum 4 panels) so that the skin effects can be seen in the output.

The panels should be bending QUAD4s and any longerons or stringers should be CBARs. Pressure loading should now load the mid-bay points.

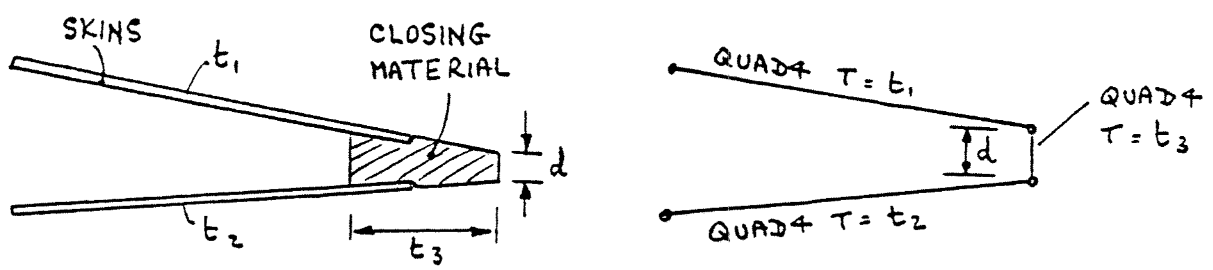
TRAILING EDGE IDEALISATION

The idealisation of trailing edges has in the past been of minor concern since for purely stressing purposes local pressure design has been critical.

With the advent of optimisation methods using aeroelastic data the trailing edge displacements are of interest, and for flutter the T/E stiffness and mass can be critical.

In reality most trailing edges do not meet at a point and it is vital to represent the real finite depth. The usual method is to include a slender QUAD4 closing element with a thickness representing the closing material. The skin elements are thus separated at the T/E and now represent the correct stiffness.

TRAILING EDGE IDEALISATION

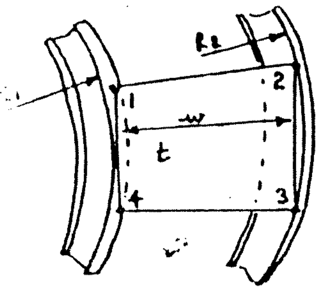


REAL STRUCTURE

IDEALISATION

FUSELAGE SKINS - COARSE MESH

SINGLE PANEL BETWEEN FRAMES - THIN SKINS



CSHEAR THICKNESS = t

AVERAGE PANEL WIDTH = w
(BETWEEN RIVET LINES)

CRODS ON EDGES 1-2 AND 4-3 REPRESENT FULL END LOAD CAPABILITY OF SKINS.

CRODS ON EDGES 1-4 AND 2-3 REPRESENT EFFECTIVE AREA OF ATTACHED SKIN - USE E.S.D.U. 71004 (SEE BELOW)

1-4 AND 2-3 ROD AREAS - SKIN ATTACHED BY SINGLE RIVET LINE.

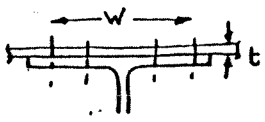
AREA ALONG EDGE 1-4 $= 0{\cdot}39\sqrt{R_1 t^3}$

AREA ALONG EDGE 2-3 $= 0{\cdot}39\sqrt{R_2 t^3}$

IF PSHEAR F2 FACTOR USED, $F2 = \dfrac{0{\cdot}39\left(\sqrt{R_1 t^3} + \sqrt{R_2 t^3}\right)}{wt}$

SKIN ATTACHED BY >1 LINE OF RIVETS

ADD $\dfrac{Wt}{2}$ TO THE EDGE CONCERNED



... W IS WIDTH BETWEEN OUTER RIVETS

THICK SKINS ... INTAKE DUCTS ETC.

EFFECTIVE SKIN AREA MAY APPROACH 100 % - REPLACE CSHEARS BY CQUAD4s ... ie FULLY END LOAD EFFECTIVE.

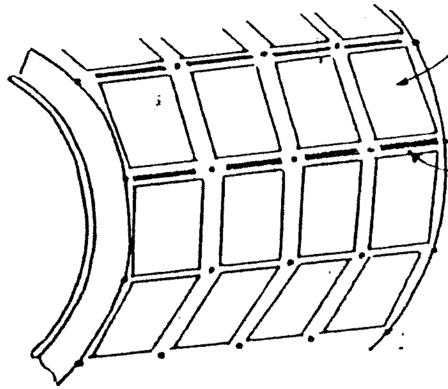
PRESSURE LOADING

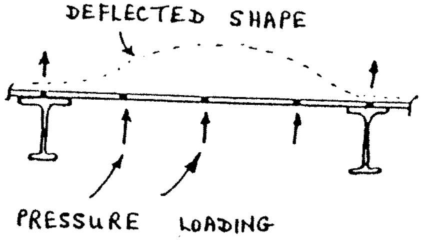
PRESSURES ON PANELS LOAD FRAME GRID ONLY

DETAIL STRESSING SHOULD COVER SKIN BENDING AND HOOP TENSION EFFECTS.

DON'T USE CQUAD4 FOR COARSE MESH
DUE TO ARTIFICIALLY HIGH END LOADS
CURVATURE ⇒ END LOAD STIFFNESS

FUSELAGE SKINS - FINE MESH





- PRESSURE LOADS NOW LOAD MID-BAY POINTS.
- CURVED SKIN EFFECTS SEEN - HOOP TENSION ETC.
- STRINGER - LONGERON LOADS IN CBARS CORRECT.
- IDEALISATION NEEDS ADDITIONAL ROTATIONAL D.O.F.
- RUNNING TIME INCREASED.

COMPOSITE PANEL IDEALISATION

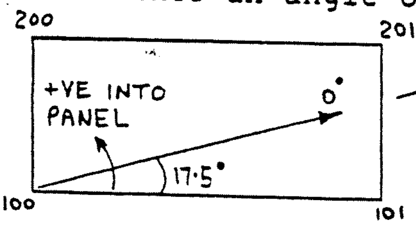
Composite panels form a large part of present day structures and several methods of idealisation are possible.

a) single QUAD4 anisitropic panel
b) mulitple QUAD4 panels
c) Nastran PCOMP facility
d) Through the thickness modelling using 2-D elements

Correct bending stiffnesses can be obtained with methods a) and c), method b) is used for membrane effects only (usually in optimisation work).

a) Single Anisitropic Panel :-

For membrane only panels the QUAD4 material is specified by a single MAT2 card, the stiffness matrix being calculated using the ADS CF02 program. The fibre angles are specified relative to the 0 deg direction, the orientation of this layer can then be input on the QUAD4 card as either an angle or a co-ordinate system, as shown below.



Cid=1

MATERIAL DIRECTION SPECIFIED AS ANGLE FROM 1-2 EDGE OR CORD SYSTEM 1

Nastran QUAD4 element would be specified as

CQUAD4	1	1	100	101	201	200	17.5
or							
CQUAD4	1	1	100	101	201	200	1

where the 1 in field 8 would point to a CORD card.

WARNING .. the material direction is the PROJECTION of the X axis onto the panel. Beware of panels out of plane with the CORD specification.

Consider a layup of total thickness 4.0mm :-

Layer	Thickness	% Thickness	1/2 unit t
+45	0.5	12.5	.0625
-45	0.5	12.5	.0625
0	2.0	50.0	.25
90	1.0	25.0	.125

CF02 should be run with layer thicknesses totalling UNITY, option 0 giving the stiffness matrix as shown below.

```
PROGRAMME CFØ2  -  ISSUE 2  -  ANISOTROPIC PLATE STIFFNESS
**************************************************************


SYMMETRIC PLANE              NO TEMPERATURE OR MOISTURE       USED
NO. OF PLIES =  8    NO. OF MATERIALS = 1    ANGL E ØF Ø DEG. DATUM =     Ø.ØØØ
MATERIAL              E1            E2             G12            NU12        MAT. NO.
M               136ØØØ.ØØØØ     6ØØØ.ØØØØ     3ØØØ.ØØØØ        Ø.3ØØØ           1
```

PLY NO.	THICKNESS	ANGLE	MATERIAL NO.	ANGLE WITH DATUM
1	0.06250	45.00000	1	45.00000
2	0.06250	-45.00000	1	-45.00000
3	0.25000	0.00000	1	0.00000
4	0.12500	90.00000	1	90.00000
5	0.12500	90.00000	1	90.00000
6	0.25000	0.00000	1	0.00000
7	0.06250	-45.00000	1	-45.00000
8	0.06250	45.00000	1	45.00000

USE UNIT THICKNESS.

INPLANE STIFFNESS MATRIX

0.79663336E+05	0.97416582E+04	0.00000000E+00
0.97416582E+04	0.47033777E+05	0.24414063E-02
0.00000000E+00	0.24414063E-02	0.10934483E+05

MAT2 CARD FOR MEMBRANE

RIGIDITY MATRIX

0.65355332E+04	0.16796389E+04	0.22305371E+03
0.16796389E+04	0.22868916E+04	0.22305360E+03
0.22305365E+03	0.22305360E+03	0.17790410E+04

~~MULTIPLY BY 12 TO OBTAIN MAT2 FOR BENDING~~

The MAT2 material data is specified using the upper triangular terms from the stiffness matrix :-

F1	F2	F3	F4	F5	F6	F7	F8
MAT2	1	79663.3	9741.66	0.0	47033.8	0.0	10934.5

The PSHELL card would thus be :-

PSHELL PID 1 4.0

If BENDING effects are to be modelled, CF02 is used with the full laminate specification in order to obtain a rigidity (bending) matrix for a UNIT thickness of laminate. The rigidity matrix is then multiplied by 12.0 and a separate MAT2 card specified for the bending material. A typical stacking sequence is shown below together with the CF02 output.

PLY NO.	THICKNESS	ANGLE	MATERIAL NO.	ANGLE WITH DATUM
1	0.06250	0.00000	1	0.00000
2	0.06250	45.00000	1	45.00000
3	0.06250	0.00000	1	0.00000
4	0.06250	-45.00000	1	-45.00000
5	0.06250	0.00000	1	0.00000
6	0.06250	90.00000	1	90.00000
7	0.06250	0.00000	1	0.00000
8	0.06250	90.00000	1	90.00000
9	0.06250	90.00000	1	90.00000
10	0.06250	0.00000	1	0.00000
11	0.06250	90.00000	1	90.00000
12	0.06250	0.00000	1	0.00000
13	0.06250	-45.00000	1	-45.00000
14	0.06250	0.00000	1	0.00000
15	0.06250	45.00000	1	45.00000
16	0.06250	0.00000	1	0.00000

USE ACTUAL STACK FOR UNIT THICKNESS

INPLANE STIFFNESS MATRIX

0.79663344E+05	0.97416602E+04	0.00000000E+00
0.97416602E+04	0.47033781E+05	0.24414063E-02
0.00000000E+00	0.24414063E-02	0.10934483E+05

RIGIDITY MATRIX

0.79856396E+04	0.11217456E+04	0.35051285E+03
0.11217456E+04	0.19525704E+04	0.35051282E+03
0.35051285E+03	0.35051279E+03	0.12211477E+04

MULTIPLY BY 12 TO OBTAIN MAT2 CARD FOR BENDING

As can be seen the rigidity matrix is quite different for the fully specified layup, and if bending effects are critical, for buckling etc., the exact stacking sequence should be used.

Multiplying the matrix by 12.0 gives an equivalent material for BENDING effects, the full thickness being used. The ratio $12I/T^3$ should be set at 1.0 on the PSHELL card.

The Bending MAT2 card would be :-

```
MAT2    2       95827.7 13461.0 4206.1  23430.8 4206.1  14653.8
```

and the PSHELL would now be

```
PSHELL  PID     1       4.0     2       1.0
```

If the MID3 field is blank the TRANSVERSE SHEAR flexibility is zero (infinitely stiff). Normally this should be specified and for CFC laminates the matrix (glue) material should be used with E=6000, say, and an effective thickness of between 0.4 and 1.0.

The final PSHELL card would be :-

$12I/t^3$

```
PSHELL  PID     1       4.0     2       1.0     3       0.8
```

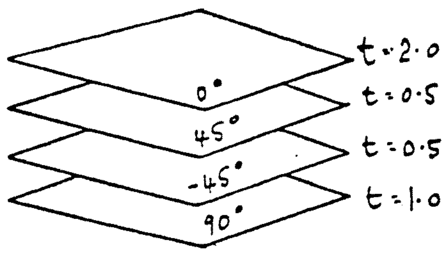
where the transverse shear material is given on a MAT1 card :-

```
MAT1    3       6000.0          0.3
```

b) Multiple QUAD4 elements

The laminate is split up into four QUAD4 elements, one for each layer direction, each having the individual fibre direction total thickness, and referencing a single MAT2 card which gives the unidirectional stiffness matrix for the material.

The main use for this type of idealisation is in optimisation work where the layer thicknesses are a variable. It is not possible to represent bending effects correctly and this work should be carried out in the detail stressing stage using the CF programes in ADS or the postprocessor.



Material angles may be specified using co-ordinate systems in the F8 of the CQUAD4, or by inserting angles from edges 1-2.

For UD material the CF02 output would be

INPLANE STIFFNESS MATRIX

0.13654216E+06	0.18071758E+04	0.00000000E+00
0.18071758E+04	0.60239185E+04	0.00000000E+00
0.00000000E+00	0.00000000E+00	0.30000000E+04

and the MAT2 card is thus

```
MAT2    1       136542. 1807.2  0.0     6023.9  0.0     3000.0
```

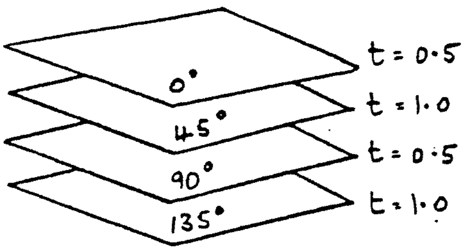
c) The Nastran PCOMP facility

Nastran has available a layered composite element property card PCOMP. The various layer thicknesses, orientations and materials are specified and the equivalent PSHELL and MAT2 cards are generated as part of the solution.

Bending and transverse shear effects are automatically included and thus a membrane only model cannot be set up using this type of data. This increases the size and complexity of the job and thus the PCOMP data is seldome used within the Stress Office.

It is possible to put in the full stacking sequence and to recover stress output for individual plies using an RFALTER - see the Nastran users handbook. The PCOMP card is referenced by the usual CQUAD4 property ID.

A typical set of data would be as shown below for a laminate with total thickness 3.0mm.



The angles on the layers are relative to the angle or Cid given on the QUAD4 card..... +ve into the panel from edge 1-2.

The material ID is 5

The PCOMP data is shown below :-

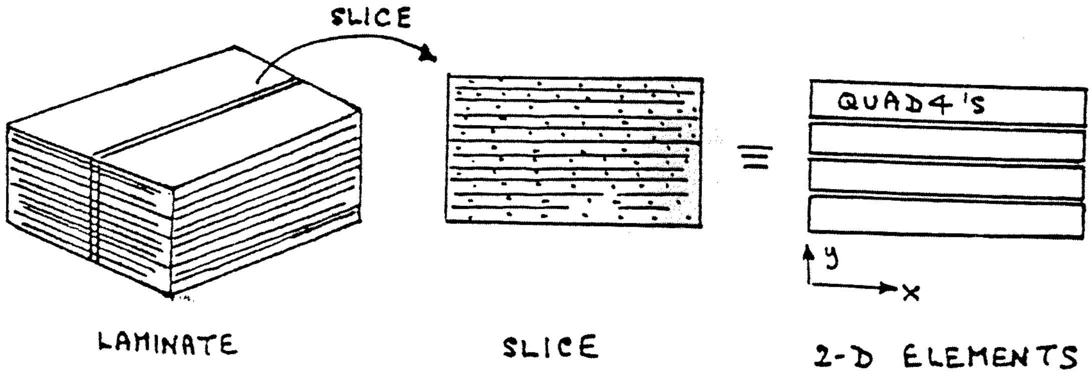
```
PCOMP    1
+81A30285        0.5     0.      YES     5       1.0   .  45.             1B1A3028
+81A30305        0.5     90.     YES     5       1.0   .  135.    YES     1B1A3030
                                                                  YES     1B1A3032
```

The computed PSHELL and MAT2 data is shown below, the three MAT2 cards being for membrane, bending and transverse shear.

```
*** USER INFORMATION MESSAGE 4379, THE USER SUPPLIED PCOMP BULK DATA CARDS ARE REPLACED BY THE FOLLOWING
PSHELL          1  100000001  3.0000E+00  200000001  1.0000E+00
      -1.5000E+00 1.5000E+00   400000001                                  0  1.0000E+00  0.0
MAT2    100000001 6.0778E+04  2.6866E+04  2.7836E-03 6.0778E+04  5.1585E-03  2.9032E+04  1.6500E-06
      0.0         0.0         0.0         0.0        0.0         0.0         0.0         0.0
                0
MAT2    200000001 8.7414E+04  2.5524E+04 -1.2647E+04 3.6825E+04 -1.2647E+04  2.7690E+04  1.6500E-06
      0.0         0.0         0.0         0.0        0.0         0.0         0.0         0.0
                0
MAT2    400000001 8.3363E+03 -2.0127E+03  6.3237E+03 -4.3110E+03 6.3237E+03 -2.0127E+03  1.6500E-06
      0.0         0.0         0.0         0.0        0.0         0.0         0.0         0.0
                0
```

d) Through thickness CFC modelling using 2-D elements.

This form of idealisation is common for investigating details of test specimens or portions of structure on a very detailed level. It is vital to know the full specification of the problem to be solved and the expected information required from the model. Boundary constraints and loading should be known and quite often symmetry can be used to reduce the size of the model. Through the thickness modelling using 2-D elements requires the determination of the material stiffess matrix where the 'through the thickness' material is the matrix (glue).



For a particular QUAD4 element the stiffness in the X direction will be dependant upon the layup stack which it represents, whilst the Y stiffness will be the glue only. Having determined Ex and Ey the usual anisotropic MAT2 data can be created by inverting the compliace (flexibility) matrix.

In more detail the method would be :-

a) determine the layups to be represented by each QUAD4 through the thickness element.

b) for each element use CF02 to produce the compliance matrix (option 0) of the group of plies (for unit thickness obviously) and deduce the value of E in the QUAD4 x direction.

c) the known material details are now
   Ex deduced X direction stiffness
   Ey glue E
   G glue G
   0.3 poisons ratio

d) calculate the element stiffness matrix by either inverting the compliance matrix formed from the data at c), or use the above data in CF02 and output the stiffness matrix for unit thickness.

e) create MAT2 data from stiffness matrix.

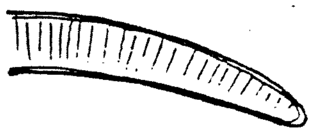
The above method is the general application for through the thickness modelling, however other aspects may require the modification of the material data before use. Poissons ratio constraints need to be accounted for if the slice is from the centre of a model as this would imply an increase in effective stiffness in the X direction. Slices at a free edge would again be treated differently. All these aspects should be discussed with the materials group prior to the creation of the model.

HONEYCOMB IDEALISATION

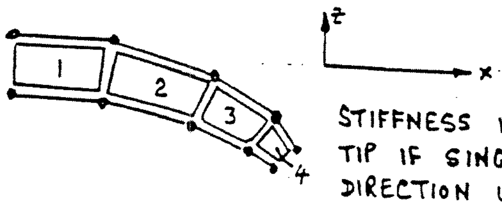
It is nesessary to model honeycomb where it is ued as the internal structure for flaps, foreplanes, fins etc. The CHEXA and CPENTA solid elements are used for this purpose and are simply defined by their corner grid points. It is possible to specify additional grid points at the centre of the edges, but this facility is not nesessary for honeycomb and is mainly used for solid modelling.

Stresses may be recovered at the centroid of the elements and at the corners, the output being in the MATERIAL direction. This can be in the global, local or element axes. For the CHEXA the element axis depends upon the SHAPE of the element (X running between the longest direction of the mid faces), whilst for the CPENTA element the it is dependant on the conectivity list. Thus it is advisable to specify the material direction in all cases via field 4 of the PSOLID card.

WARNING ...since honeycomb is extreemly anisotropic, make sure that the material direction follows the manufacturing drawings. If a single direction is used for a thin curved panel the out of plane stiffness will be lost as the elements curve away.



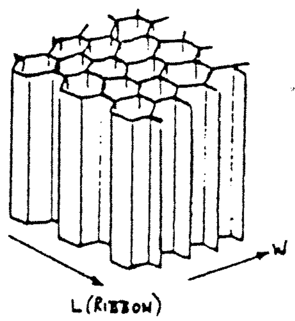
LEADING EDGE SECTION

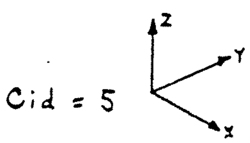


STIFFNESS IS LOST TOWARDS TIP IF SINGLE MATERIAL DIRECTION USED

The material can be specified using a MAT9 card, which holds the 6x6 stiffness matrix. Take care with the data to ensure that the matrix ties up with the material axis system, ~~ie.~~ which planes are the L and W directions.

A typical honeycomb PSOLID, stiffness matrix and MAT9 card would be as shown below.





Honeycomb stiffness matrix

X	6.5						
Y		5.5					
Z			2300				
XY				1.0			
YZ					1149		W
ZX						1293	L

PSOLID ID 15 5

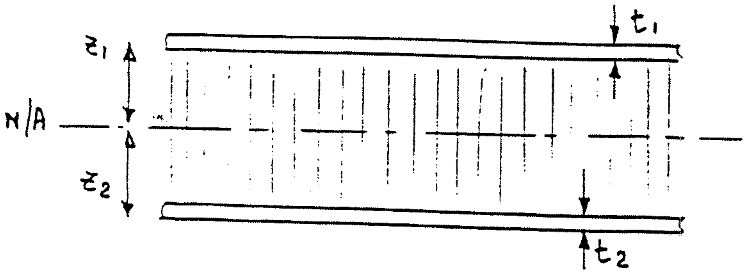
Where the material ID is 15 and direction is in coordinate system 5.

F1	F2	F3	F4	F5	F6	F7	F8	F9
MAT9	15	6.5	0.0	0.0	0.0	0.0	0.0	5.5
+	0.0	0.0	0.0	0.0	2300.	0.0	0.0	0.0
+	1.0	0.0	0.0	1149.	0.0	1293.		

The L and W stiffnesses are obtainable from the usual data sheets, the crushing (z) stiffness possibly. The x,y and xy values are small due to the concertina effect, and may be inserted as 1.0 if unknown.

SANDWICH PANELS

These can be modelled explicitly with elements for the two skins and internal core, or implicitly where a single bending element represents the assembly of the three componants. The method for the latter case is shown below.



NOTE:- Z1 AND Z2 MUST BE SPECIFIED RELATIVE TO THE ELEMENT LOCAL AXIS

Set the PSHELL membrane thickness to t1+t2 (if fully effective) with the associated material MID1.

Calculate the section moment of inertia, I

Calculate the ratio $K = 12I/t^3$ where $t = t1+t2$

Assume the bending is taken as skin loads, put MID2 = MID1 and K into the PSHELL fields 5 and 6

The transverse shear material MID3 must be specified based on the shear stiffness of the core and the ratio ts/t, where ts is the transverse shear thickness.

The full PSHELL card is thus :-

PSHELL	PID	MID1	t	MID1	K	MID3	ts/t
			membrane		bending		transverse shear

The Nastran output gives the correct skin loads if Z1 and Z2 are specified correctly on the PSHELL continuation card.

A typical PSHELL card may look like this

PSHELL	5	1	3.0	1	250.0	2	10.0

This would show that the sandwich is 250 times stiffer in bending than the 3.0mm skin and that the core thickness is 30.0mm.

JOINTS

The manufacture of aircraft componants involves all kind of joints. In the idealisation process the majority of these can be ignored in that they are the continuous bolted/rivetted/bonded joints which, although subject to local stressing problems - shear build up etc., do not significantly affect the stiffness of the structure as a whole.

The remainder of the joints can be modelled explicitly - by representing each bolt, lug etc. in some detail, or implicitly by modifying the local element properties or grouping the effects of several joints/fasteners to pairs of co-incident grids and using spring elements to represent the equivalent stiffness. If the element property is modified, the derived loads will be applied to the real structure sizes, and fastener loads will be calculated from the edge loads on the panel. If spring elements are used the fastener loads are known and the loads in the panel (real thickness) too.

One of the problems for bolted or rivetted joints is the derivation of the individual or group bolt stiffness. Once these have been determined it is a simple task to create the spring element values. Early work at BAe Warton produced reports on the stiffness of bolted and rivetted joints (SOR(P) 75 etc.) and these have now been incorporated into ESDU 85034 and 85035.

Explicit modelling of joints can be simple, ie. for transport type joints a single bolt is used at each position, or complex, ie. a wing - fuselage fitting involving many bolts. Care should be taken to ensure that any load offset effects can be carried by the local structure and that unreal flexibilities are not created. It is better to move a bolt position to a local 'hard' point on the model and ensure the overall stiffness is maintained, the local stressing taking into account the real bolt positions and the offset effects.

The modelling of large lug and pin assemblies in some detail is part of the design process and if carried out correctly can predict the overall joint stiffness, which can then be used as a simple spring element in the overall coarse mesh model.

HINGES

Hinges can be modelled by

a) using common grids between componants with only translational D.O.F., ie. rotations are possible.

or b) using co-incident pairs of grids and connecting these only in certain translational D.O.F. (CELASi or MPCs).

Method b) is preferrable in that the effects of sliding fittings can be modelled by only connecting the pairs in certain directions. If a structure is connected by several hinges it is important to make these co-linear, the best method being to use a coordinate system for the geometry of the hinge line points. Try and make the hinge line direction follow the nearest basic x, y, or z direction.

WARNING .. the pairs of coincident grids MUST have identical global D.O.F.'s. If this is not so the spring elements will transform the displacements (and loads) across the pair of grids and the resulting structure will be out of balance. This implied transformation is occasionally used to advantage for setting flaps at angles etc. but in general the above comment holds.

WARNING .. if the hinge system is redundant make sure that the geometry of the hinges is colinear, using a hinge line co-ordinate system if necessary. If this is not so the hinges will lock up and present a hinge moment load path which is unreal.

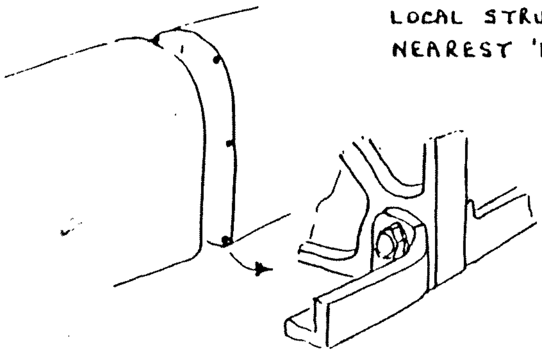
MECHANISMS

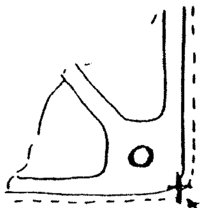
These can be modelled using ROD or RROD elements to represent the links, jacks etc. Nastran will calculate the displacements under load, but the loads are calculated with the geometry at the undeflected position (unles large displacement solution is used). Thus beware of exessive deflections.

JOINTS – EXPLICIT MODELLING

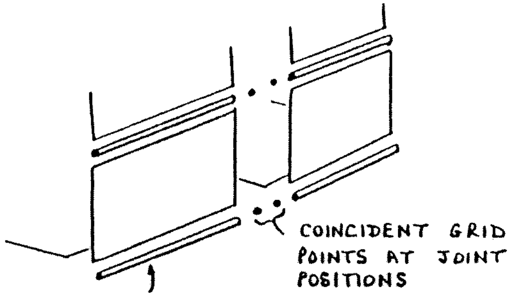
TRANSPORT TYPE JOINT

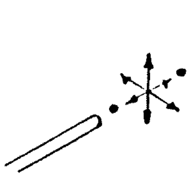
UNLESS DETAIL MODEL CAN REPRESENT LOCAL STRUCTURE – MOVE CONNECTION TO NEAREST 'HARD' POINT.





GRID AT SKIN LINE

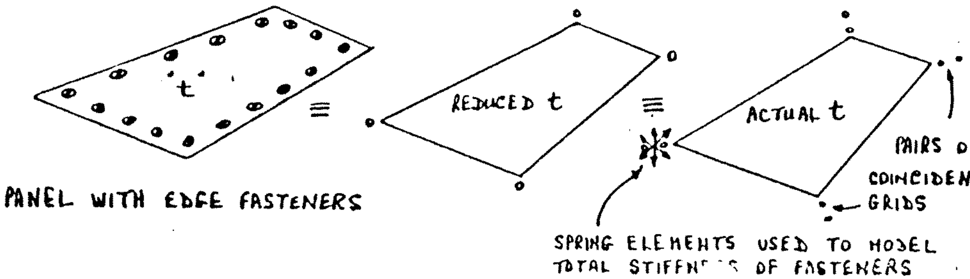




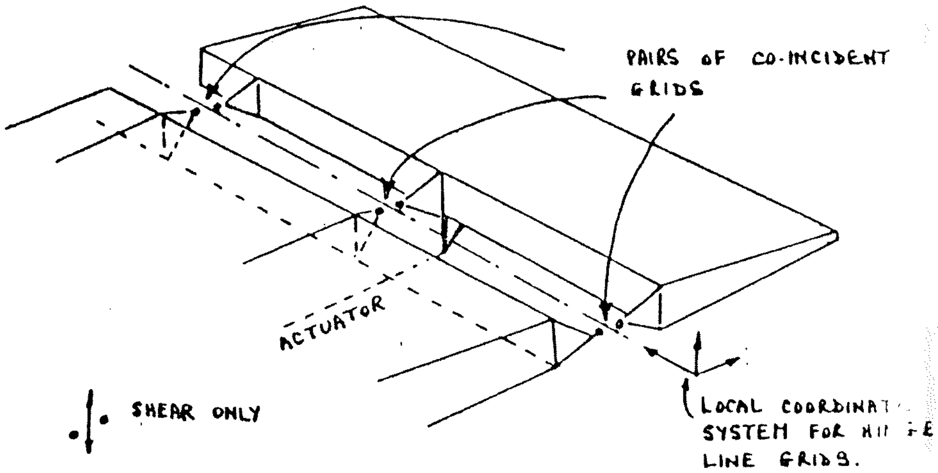
CORRECT SPRING LOADS TO ACTUAL JOINT POSITIONS – MAINTAINING FORCE BALANCE. DETAIL STRESS LOCAL ATTACHMENT AREAS

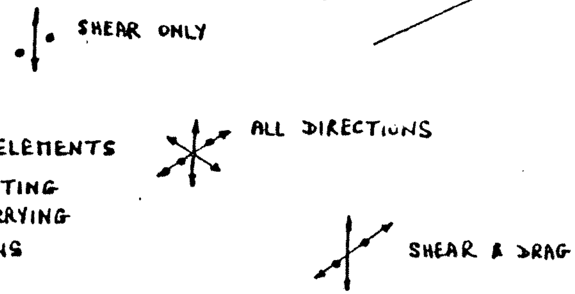
IMPLICIT MODELLING

THE EFFECTS OF THE JOINT ARE IMPLIED BY MODIFYING THE ELEMENT SIZE OR COLLECTING THE EFFECTS TO THE ELEMENT GRIDS.



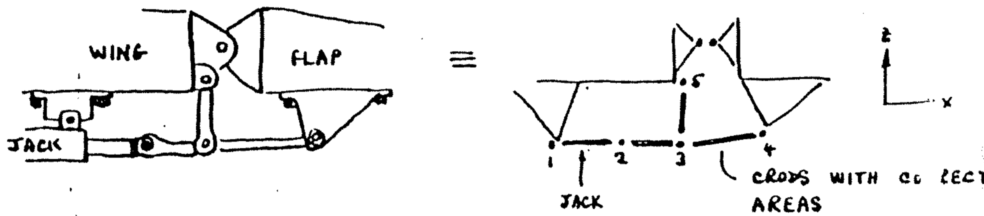
HINGES





THE GLOBAL (FIELD 7) AXIS SYSTEM FOR THE PAIRS OF GRIDS IS SET TO THE LOCAL HINGE LINE SYSTEM. SPRING ELEMENTS ARE THUS ORTHOGONAL TO THE HINGE LINE.

MECHANISMS – THESE CAN BE REPRESENTED WITH CRODS OR RBDs.



NOTE:- POINT 2 WILL BE SINGULAR IN THE Z DIRECTION.

LOADS IN MECHANISM LINKS ARE CALCULATED WITH GEOMETRY AT ORIGINAL POSITION. – UNLESS LARGE DISPLACEMENT THEORY USED.

HOLES AND REINFORCEMENTS

The treatment of these on a coarse mesh idealisation is to modify the element size to give an equivalent stiffness. The criteria to be used is whether or not the local features will have any noticable effect beyond the boundaries of the element in which they occur. Features such as repeated stiffeners on a skin or a series of lightening holes in a web...items which occur regularly, cannot be ignored.

AGARD lecture series 147 included papers given by I.C.Taig dealing with the practical application of finite element analysis, and he catorigises the guide shown below.

Feature Type	Can be ignored if:-
Reinforcing features	Reinforcement is not continuous AND Aggregate volume of reinforcements <10% of basic element volume
Weakening features	No more than 20% section lost in any continuous loadpath across the element AND Aggregate volume of perforations <5% of basic element volume
Joints	Aggregate flexibility over periphery of element <10% of element flexibility under relevant uniform loading

These rough rules are intended to ensure that the strain energy in the element with its features is within $\pm$ 10% of the basic element under any relevant loading.

For weakened shear webs the equivalent thickness and end load area is shown in the table below (from AGARD series ~~147~~).

Equivalent Stiffness of Weakened Shear Webs

Type of Web	Effective Shear Stiffeners (equivalent Gt)	Effective area A of web associated wih flange: Beams in flexure	Effective area A of web associated wih flange: Panels with low stress gradient
Plain webs	Gt		
Honey comb sandwich webs	$\sum$ Gt for skins	$\frac{bt}{6}$	$\frac{bt}{2}$
Shear-buckled plain web	0.6 Gt	Lesser of $15t^2$ or bt/6	$15t^2$
Web with lightening holes	Gt $(1-D/_{s})$	$\frac{bt}{6}$	$\frac{(b-D)t}{2}$
Corrugated webs	Gt (a/a_d)	Zero normal to corrugations	Zero normal) to $\frac{bt}{2}$ parallel) corruga-)tions
Web with shallow swages	Gt	Lesser of $15t^2$ or $\frac{bt}{6}$	$15t^2$ normal) to $\frac{bt}{4}$ parallel) swages
Castellated webs	$\dfrac{Gt}{1+\frac{\alpha\beta}{(1-\alpha)}+\frac{0.4\beta^2\gamma^2}{(1-\alpha)^3}}$	Zero along line of castellations	zero along) to line $\frac{bt}{3}$ normal) of) castella- tions

where:- a, b,t = web dimensions $\qquad a_d$ = developed length $\qquad$ D hole diameter

$\alpha = \frac{\text{notch width}}{\text{pitch}} \qquad \beta = \frac{2 \times \text{notch depth}}{\text{plate depth}} \qquad \gamma = \frac{\text{plate depth}}{\text{notch pitch}}$

Reinforcements can be modelled by increasing the overall thickness or, in the case of stiffeners in a single direction, by adding CRODs on the panel edges in the stiffener direction. Alternatively the element material E value can be adjusted to reflect the stiffness change.

Wherever an element is changed in stiffness, the interpretation of the results should originate from the element LOADS which are then applied to the real structure for detail stress calculations.

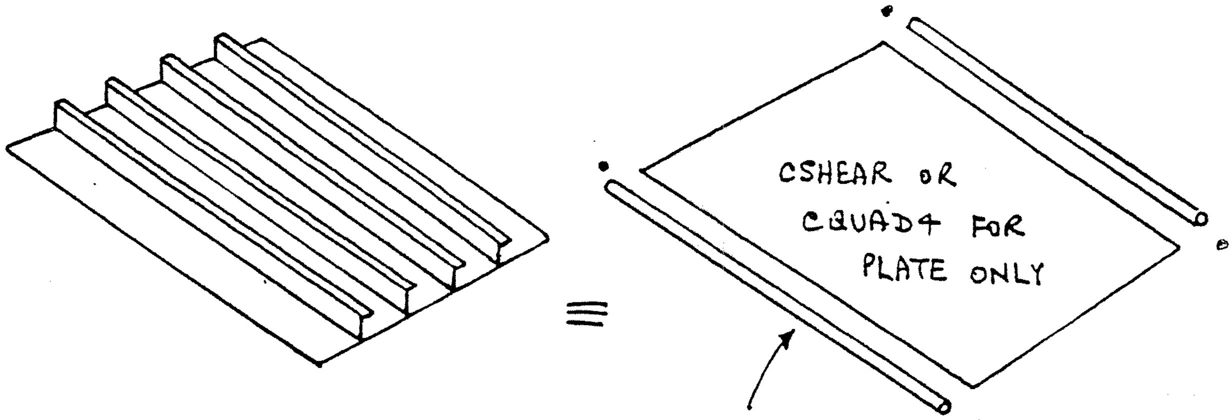


PLATE WITH STIFFENERS.

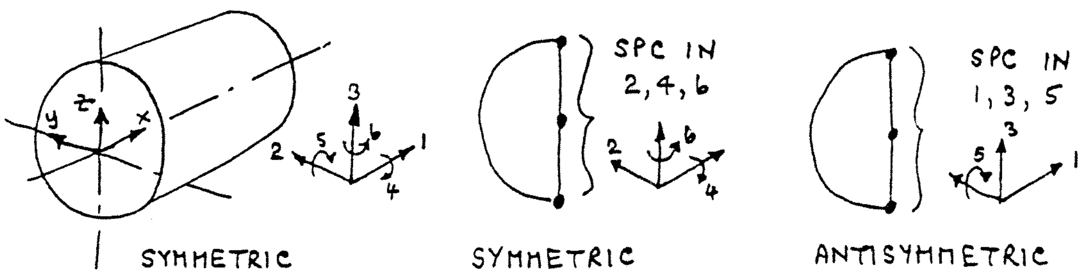
ADDITIONAL CRODS REPRESENTING STIFFENER AREAS.

SYMMETRY

Symmetry can be used to simplify the model and thus reduce the runing time, or enable a finer mesh to be set up in a given time. There are various considerations which should be taken into account when determining its use.

a) is the structure geometry symmetric
b) how many planes of symmetry are present
c) does the material behave symmetrically.. CFC ?
d) are the constraints symmetric
e) is the loading symmetric .. it is possible to break down any loading into symmetric and antisymmetric parts, which can then be applied to the model as seperate cases with different constraints and added. Some users would prefer not to do this for small models.

If symmetry is to be used, only a portion of the structure is modelled, and the grid points on the plane or planes of symmetry have imposed on them constraints, SPCs etc. Two kinds of symmetry are used, symmetric and antisymmetric, the most common use being for fuselages as shown below.



SYMMETRIC STRUCTURE | SYMMETRIC CONSTRAINTS | ANTISYMMETRIC CONSTRAINTS

For symmetric loading the grids on the ZX plane should be constrained in the 2,4 and 6 freedoms, ie. preventing lateral, roll and yaw movements. For the antisymmetric case freedoms 1,3 and 5 are constrained, preventing aft, vertical and pitch movements.

These type of constraints can be applied to any structure to FORCE symmetric or antisymmetric BEHAVIOUR. A table of the constraints required for the various planes of symmetry is shown below.

Symmetric constraints :-

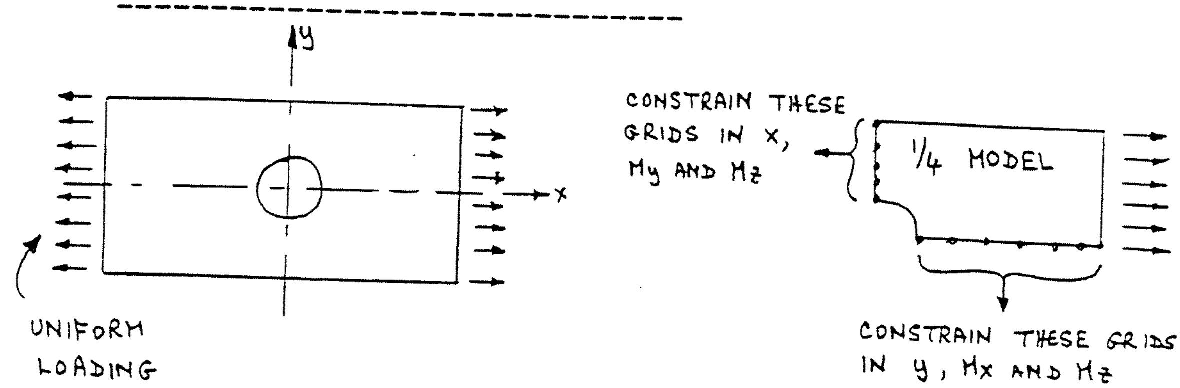
Plane of symmetry	X	Y	Z	MX	MY	MZ
XY			Fix	Fix	Fix	
YZ	Fix				Fix	Fix
ZX		Fix		Fix		Fix

Antisymmetric constraints :-

Plane of symmetry	X	Y	Z	MX	MY	MZ
XY	Fix	Fix				Fix
YZ		Fix	Fix	Fix		
ZX	Fix		Fix		Fix	

Two further examples are shown below :-

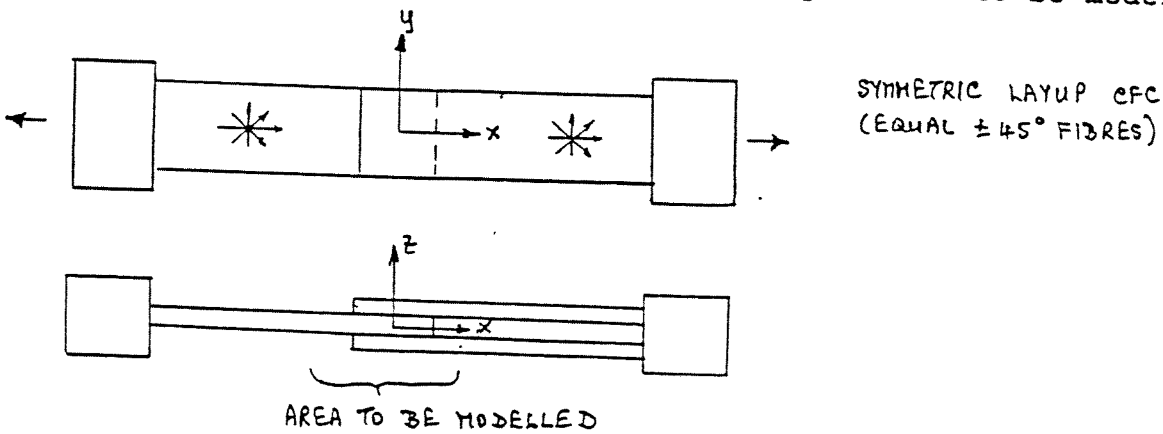
Simple test specimen with hole



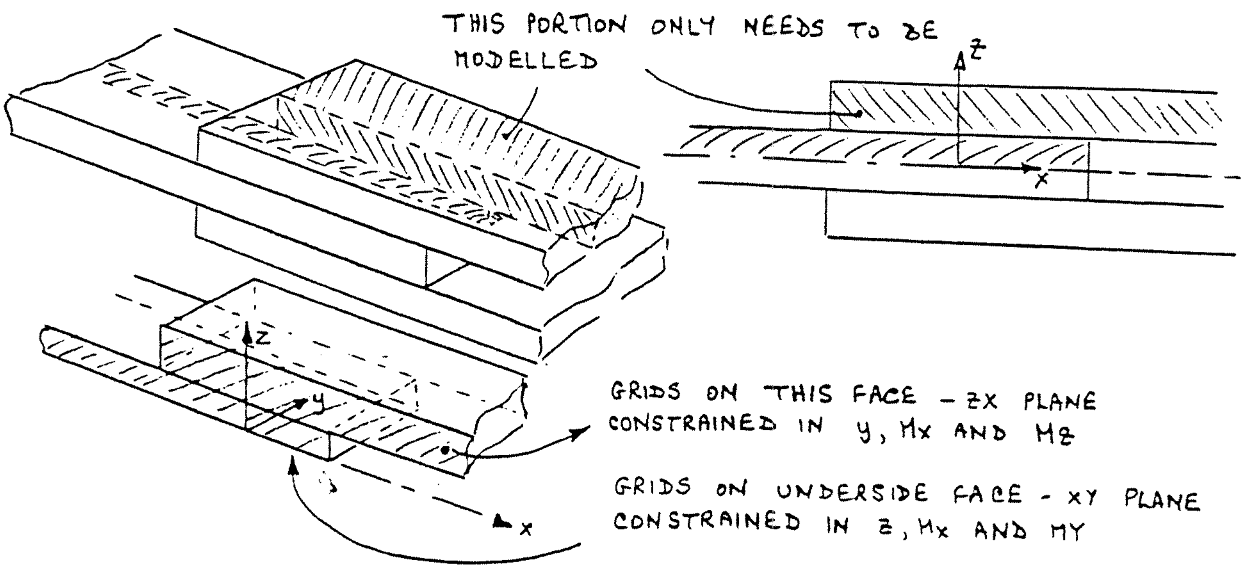
There are two planes of symmetry for SYMMETRIC conditins, ZX and YZ, and the model can be reduced to 1/4 size.

Bonded test joint

Symmetric lap joint using symmetric layups in CFC and axial loading. The centre section of the joint is to be modelled.



There are two planes of symmetry for SYMMETRIC conditions, XY and ZX, and the model can be reduced to 1/4 size.



SOLID MODELLING

By definition this is the representation of the structure using three-dimensional elements, CHEXA, CPENTA etc. A thick plate previously modelled using bending CQUAD4s is now defined by solid elements with grid points on the both surfaces of the plate. Thus the geometry mirrors the exact geometry of the item.

No rotational D.O.F. are present at the grid points since by definition bending loads are carried through the elements depth.

It is only possible to recover element STRESSES at the centriod and at the corner grid points. GPFB's will obviously give the corner forces acting on the grids.

Solid elements can be used in a variety of situations :-

a) large machined items, the geometry being obtained from Anvil or Catia. This is specialised analysis work and is rarely undertaken. The EFA foreplane spigot frame is an example where the Catia model is being converted to an equivalent Nastran model.
b) macro/micro investigations of detailed areas, holes, test specimens, carbon fibre layers/interlaminar features etc.
c) as core elements for flaps, fins etc.

One of the drawbacks with solid elements is that the stress levels for the adjacent elements at the same grid point have to be averaged to find the correct corner stresses. Thus the plotting or calculation of stresses by hand is difficult. Patran performes this averaging in the contour plotting routines, and this is the usual method of gaining an overall picture of the surface stress levels. A method of finding the correct surface stresses is to use strain gauge elements, see below.

STRAIN GAUGE ELEMENTS

These are normal Nastran elements, CQUAD4, CROD etc., which have insignificant thicknesses or areas and are "smeared" over the surfaces and edges of a solid element model in order to register surface stress levels. Thus more information than is usually available from solids can be recovered (mid face stress levels) and plotting is enhanced.

RIGID LOAD PATHS

Rigid load paths or elements do not occur in real life. However there are many instances in finite element modelling where it is necessary or essential to use these.

a) to remove near singularities due to the method of modelling.

b) to enforce local displacements on singular points.

c) to represent very stiff items of structure.

d) change in mesh size, joining different elements.

e) to obtain average motions of a set of points.

f) to distribute loads to a structure.

g) changing the global D.O.F. at a grid point

Rigid elements are covered in the Handbook for Linear Analysis section 2.5.4 and several applications are described in the Applications Manual section 2.10.

WARNING .. all applications of rigid elements require the use of independant and DEPENDANT freedoms. Care should be taken to ensure that the dependant freedoms (UM set) do not conflict, with each rigid element or with other modelling requirements, SPCs, boundary points etc. Freedoms from which stiffness is to be derived should not appear in the S set (Singular).. these are not necessarily only the independent freedoms.

Examples of the above types are described below

a) Removal of potential singularities.

AUTOSPC will remove EXACT and VERY NEAR singularities, but it is better to include these on the GRID cards in field 8, or by the use of the GRDSET card which applies constraints to ALL the grids in specified global D.O.F. However near singularities cannot be automatically dealt with and the user must remove thes with rigid elements. Some common situations are described below.

1. Unsupported skin panels. Where the mesh is finer than the modelled internal structure and the skins do not have bending stiffness, the unsupported grid points will have a very low stiffness out of plane. If not constrained, large local displacements will be seen (several metres), and the local skin membrane stiffness will be reduced.
Using an RBE1 the unsupported D.O.F. only is 'beamed' to adjacent hard points. The resulting loads on the three points should be insignificant, these can be seen in the GPFB for the grids concerned as the out of balence loads.

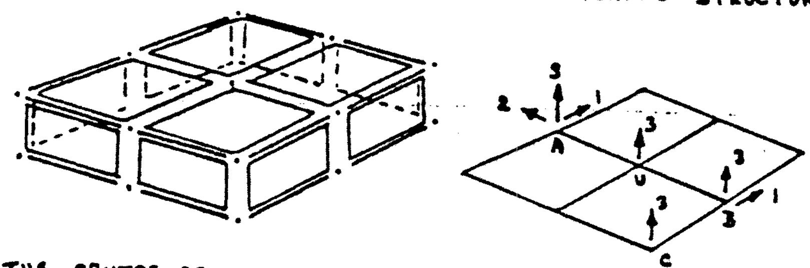
2. CSHEAR panel singularities. In modelling shear webs for wing spars or ribs etc. the CRODs along the flanges cover the end load singularity in these directions, but through the wing depth the end load capability is usually neglected. The singularity can be removed by linking the top and bottom grids with a RROD in the through the thickness D.O.F.

3. The removal of high in plane rotations. Nastran plate elements, in common with many finite element packages, do not support the in plane stiffness at the grid points. Thus there are implicit singularities for these elements, and usually for flat surface models using bending elements these are removed automatically. However if the surface is slightly curved the in plane stiffness will be made up of small components of the plate bending stiffnesses. AUTOSPC will not remove these and thus high rotations may be generated. These rotations may be removed using RBE1 elements linking the offending freedoms to local in plane freedoms.

a) REMOVAL OF POTENTIAL SINGULARITIES

1. UNSUPPORTED SKIN PANELS - RBE1

CENTRE GRID IS UNSUPPORTED BY INTERNAL STRUCTURE

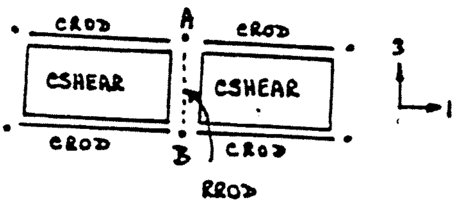


THE CENTRE GRID U IN DIRECTION 3 IS MADE DEPENDANT ON GRID A 1,2,3 , GRID B 1,2 AND GRID C 3.

RBE1 ID A 123 B 13 C 3
+ UM U 3

DEPENDANT GRID AND FREEDOM

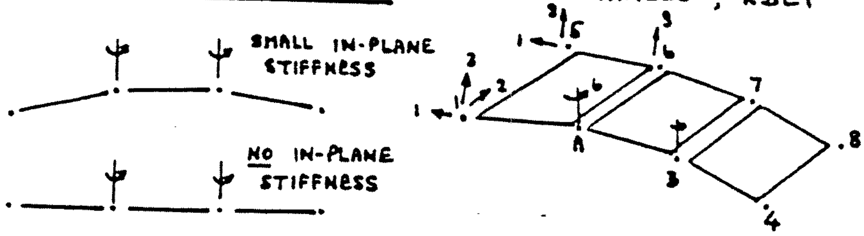
2. CSHEAR PANEL SINGULARITIES - RROD



GRID A IS LINKED TO GRID B BY A RIGID RROD ELEMENT

RROD ID A B 3 .... GRID A FREEDOM 3 IS DEPENDANT

3. HIGH IN PLANE ROTATIONS - BENDING PANELS , RBE1



RBE1 ID 1 123 5 13 6 3
+ UM A 6

b) Enforcing local displacements.

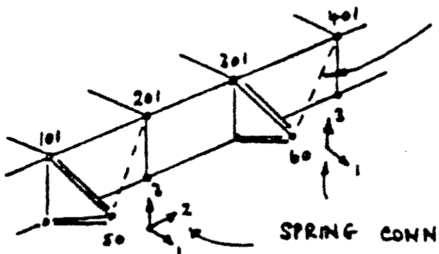
It is sometimes necessary to force a LOCAL displacement onto a singular point in order to set up a load path not modelled in adequate detail. In this case the rigid element will carry load if any load exists in its direction, but make sure that only ONE load path is present.

Brackets modelled in 2-D membrane only form usually are singular out of their plane, and RRODs may be used to transfer out if plane loads to the main structure. An RROD links the bracket point to a grid on the main structure with the specification using one of the global components in that direction. The RROD will maintain a FIXED distance between the two grid points used. Several brackets may be treated in this way, but beware of setting up redundant load paths, see below.

WARNING ... The application of enforced displacements will lead to erroneous results if a closed load path is formed. Thus if two structures are attached via brackets with potential singularities removed using RIGID elements, and more than one load path is present THROUGH the rigid elements, the resulting relative displacements may give rise to unreal loads in the structure.

b) ENFORCED LOCAL DISPLACEMENTS

HINGE BRACKETS



RRODS CONNECTING GRID 50 TO 201 AND 60 TO 401 IN FREEDOM 2.

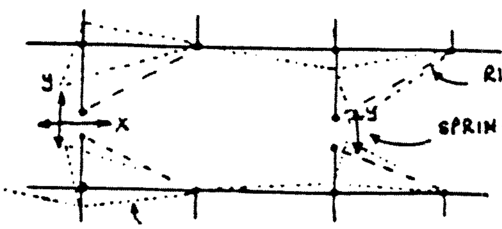
NOTE :- MAINTAINS FIXED DISTANCE BETWEEN GRIDS

SPRING CONNECTIONS TO STRUCTURE - ONLY CONNECTED IN DIRECTION 2 BY ONE SPRING

RROD	ID	201	50	2	GRID 201 FREEDOM 2 DEPENDANT
RROD	ID	401	60	2	" 401 " 2 "

WARNING

a) NON-REDUNDANT RIGID ELEMENTS IN X DIRECTION.



RIGID ELEMENTS

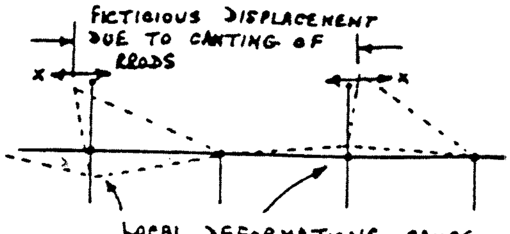
SPRING CONNECTIONS

DEFLECTED SHAPE

SPRING LOADS IN X DIRECTION WILL BE INDEPENDENT OF LOCAL DISPLACEMENTS

b) REDUNDANT RIGID ELEMENTS IN X DIRECTION

FICTICIOUS DISPLACEMENT DUE TO CANTING OF RRODS



SPRING LOADS IN X DIRECTION WILL BE "FICTICIOUS".

LOCAL DEFORMATIONS CAUSE "SPACE FRAMEWORK" MOVEMENT OF RRODS

c) Very stiff items of structure.

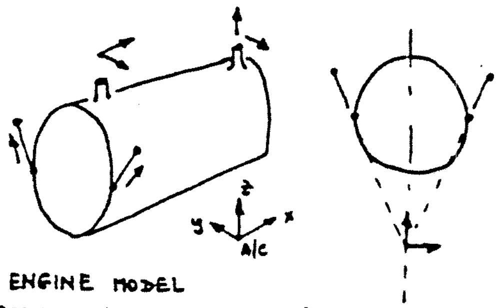
Because Nastran has to form and invert the stiffness matrix, any effects in the modelling which spoil the matrix conditioning should be avoided. One of these is the inclusion of very stiff elements, which make the conditioning poor because of the increase in the numerical range of the stiffness matrix terms.

Very stiff items such as engines, U/C legs, heavy brackets etc. will cause this ill-conditioning if modelled, and can lead to quite implausable results. (Usually noticed by high values of Epsilon and diagonal ratio terms).

It is better to represent these items using rigid elements. Thus an engine connected to the structure in a statically determinate manner can simply be represented by a RBE1 linking the CG point to the attachment points.

c) <u>VERY STIFF ITEMS OF STRUCTURE</u>

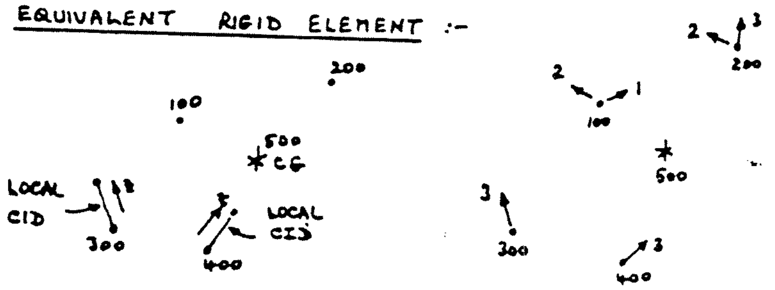
DO <u>NOT</u> ATTEMPT TO MODEL THESE - USE RIGID ELEMENTS



FORWARD LINKS FORM 2 D.O.F.
- LINKS SET UP WITH LOCAL COORDINATE SYSTEMS Z <u>ALONG</u> LINKS.

ENGINE MODEL
(STATICALLY DETERMINATE)

<u>EQUIVALENT RIGID ELEMENT</u> :-



<u>GRIDS</u> <u>STATICALLY DETERMINATE FREEDOMS</u>

```
RBE1   ID   100   12       200   2 3   300   3
+           400   3
+      UM   500   123456
```

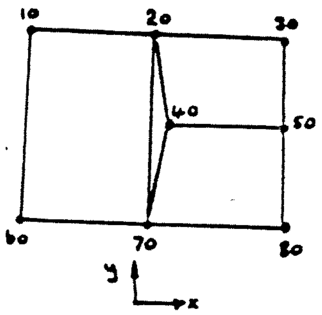
d) Merging a change of mesh and joining different elements.

Merging a change of mesh size can be accomplished using the usual modelling techniques, but sometimes this is not convenient and additional points have to be added directly as mid points along an existing element side. It is not recommended to use these higher order elements, and a rigid element should be used to enforce an average displacement on the grid concerned. Do not use this technique in areas of rapidly changing loads.

Instances occur where dissimilar elements have to be joined and continuity of load paths created using rigid elements. A common example is a bending plate joining a solid element. If the plate locates to one face of a CHEXA a single RBE1 can transfer the missing rotational load paths as differential loads at the CHEXA corners.

A more complex merge of a plate to the centre of the CHEXA could transfer the shear as the average of the corners using MPCs, and pairs of RBARs to carry the bending loads to the corners.

d) CHANGE IN MESH SIZE



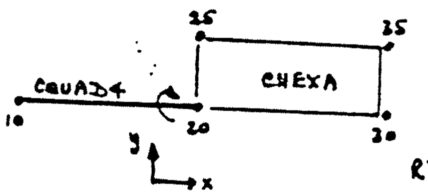
POINT 40 IS INTRODUCED TO CHANGE MESH SIZE.

STIFFNESS IN Y DIRECTION IS ACCEPTABLE.

STIFFNESS IN X DIRECTION IS TO BE AVERAGED FROM GRIDS 20 AND 70

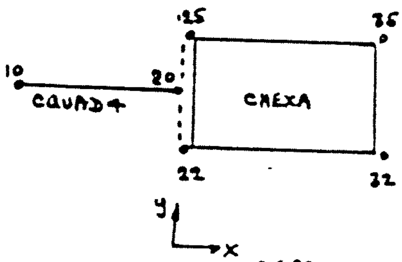
RBE1 ID 20 123 70 13 60 3
\+ UM 40 1

JOINING DISSIMILAR ELEMENTS



BENDING ROTATIONS FROM QUAD4 AT GRID 20 IS APPLIED TO CHEXA USING RBE1

RBE1 ID 20 123 25 13 35 3
\+ UM 20 6



USE MPC TO AVERAGE Y AT GRID 20 TO GRIDS 22 AND 25

USE RBARS TO PUT QUAD4 BENDING TERMS ONTO GRIDS 22 AND 25

DEPENDENT

MPC ID 20 2 1·0 25 2 -0·5
\+ 22 2 -0·5

INDEPENDENT

RBAR ID 20 25 123456 _ _ 13
RBAR ID 20 22 123456 _ _ 13

DEPENDENT

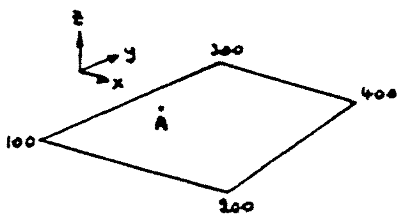
e) Obtaining average motions.

The RBE1 element is capable of defining the motion of a free point to be dependent upon a statically determinate set of motions of selected grids and freedoms. One of its uses is in the extraction of the motion of aeroelastic points not present in the model. Another RBE1 use is in the definition of interface points between structures where the points are not part of the local model. Take care that over flexible points are not used in the definition of the element, otherwise incorrect interface loads may be generated.

The RBE3 element allows the average motions of any number of points to be enforced on a specified point. The normal use of this rigid element is to predict the average motion of a fuselage frame, by first selecting a subset of 'hard' points, and secondly creating a new grid at the A/C C/L. The RBE3 element then links the motion of this new grid point to the average of all the 'hard' points. A scalor can be used for for the displacements of any of the selected points prior to the averaging, thus soft points can be made less effective.

e) OBTAINING AVERAGE MOTIONS

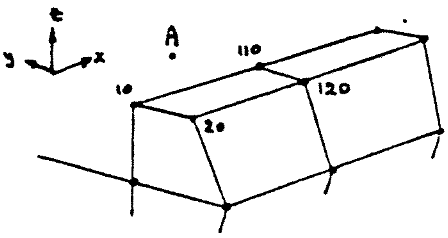
STATICALLY DETERMINATE MOTIONS



AEROELASTIC INTERPOLATION FOR POINT A

Z DISPLACEMENT OF POINT A IS THE LINEAR AVERAGE OF THE Z DISPLACEMENTS OF POINTS 100, 300 AND 400.
IF POINTS 100, A AND 300 ARE COLINEAR, POINT 400 HAS NO EFFECT ON DISPLACEMENT OF A.

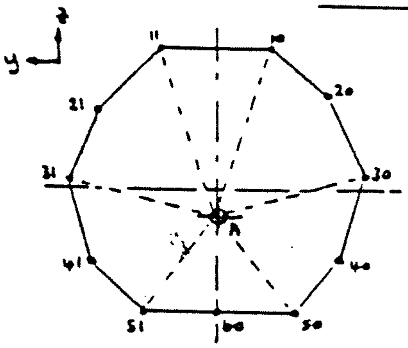
```
RBE1  ID   100  123  300  13  400  3
+     UM   A    3
```



ENSURE THAT 'HARD' POINTS ARE USED WHEN INTERPOLATING FOR DISPLACEMENTS OF A.

ATTACHMENT POINT A

```
RBE1  ID   20  123  10  13  120  3
+     UM   A   123
```



POINT A HAS ASSIGNED TO IT TH AVERAGE DISPLACEMENTS OF 6 GRID POINTS ON THE FRAME

SCALING FACTOR

```
RBE3  ID   A    123456  1.0  123  10  11
+     20   31   50      51
+     UM   10   123     31   13   50  2
```

NOTE :- THE UM SET MAY BE DELETED IF POINT A IS ALLOWED TO BE DEPENDENT.

AVERAGING MOTIONS

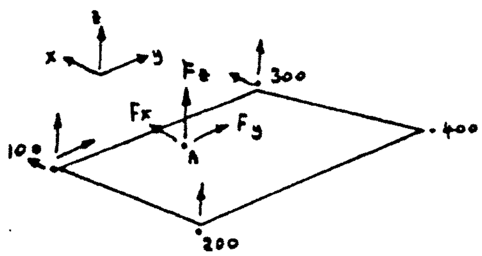
f) Distributing loads to a structure.

Loading defined at a point not coincident with any grid point may be applied by setting up an additional grid at the load point and using rigid elements to beam the load to the required grid points.

A simple beaming to 3 points is carried out using an RBE1 with the loads being applied to the model in the way the statically determinate set is defined.

An RBE3 element can be used to distribute load from a fuselage generalised point to a selected set of loads on a frame, the distribution being in the classic bolt group type of calculation. This method is used to load fuselages in coarse loading situations.

f) DISTRIBUTING LOADS TO A STRUCTURE



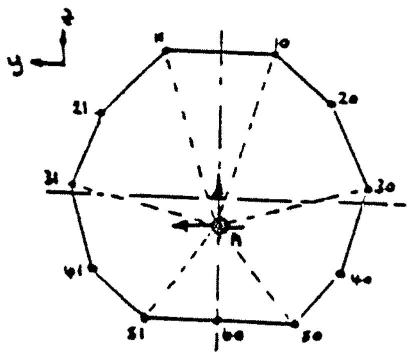
SIMPLE BEAMING OF LOAD AT POINT A USING RBE1

VERTICAL LOAD F_z WILL BE APPLIED TO GRIDS 100, 200 AND 300.

HORIZONTAL LOAD F_x WILL BE APPLIED TO GRID 100, WITH DOZ. MOMENT TAKEN AS ± x LOADS ON 100, 300.

LATERAL LOAD F_y WILL BE APPLIED TO GRID 100, WITH D.O.Z. MOMENT TAKEN AS ± x LOADS ON 100, 300.

```
RBE1  ID   100  123  200  3  300  13
+     UM   A    123
```



LOADS APPLIED TO FRAME USING GENERALISED POINT A

LOADS APPLIED TO POINT A WILL BE APPLIED TO THE SELECTED GRIDS IN THE USUAL 'BOLT GROUP' METHOD.

```
RBE3  ID  A   123456  1.0  123  10  11
+     20  21  50      51
+     UM  10  123     31   13   50  2
```

NOTE :- THE UM SET MAY BE DELETED IF POINT A IS ALLOWED TO BE DEPENDENT.

g) Changing the global D.O.F. at a grid point.

The global D.O.F. is defined in field 7 of the grid data and thus all displacement related data will be in whatever co-ordinate system is contained in this field. Sloping frames etc. may have the geometry and the global D.O.F. defined in a local co-ordinate system.

It is sometimes required that the output displacements need to be known in a different displacement system to global, and the latter cannot be changed for other reasons. This may occur at hinges where the hinge line connecting springs dictate local global D.O.F., but the displacements relative to the main structure are of interest.

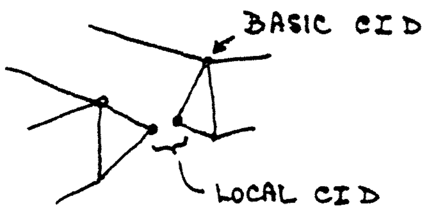
By setting up coincident grids with the required GLOBAL D.O.F. and connecting them to the original grids using RBARs the new grids will output the displacements required. Thus the RBAR carries out the transformation between the pairs of coincident grids.

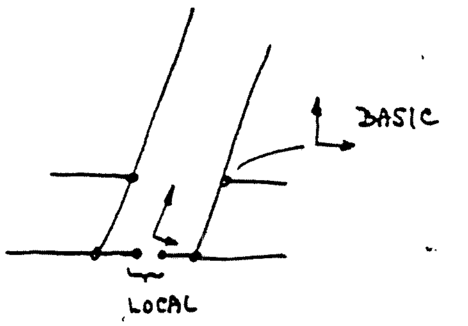
Another use would be in the speciøfication of MPC data which for geometrical reasons needs to be in a consistant co-ordinate system. New grids are created coincident with the existing grids and connected by RBARs. The MPCs can then be applied to the new grids in a consistant manner. This is a common problem involving spring elements also, the user must ensure that the pairs of grids are in the same global system.

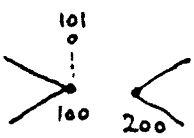
GRID ID CID X Y Z CD PS

CID ↑ GEOMETRY DEFINITION

CD ↑ GLOBAL DEFINITION



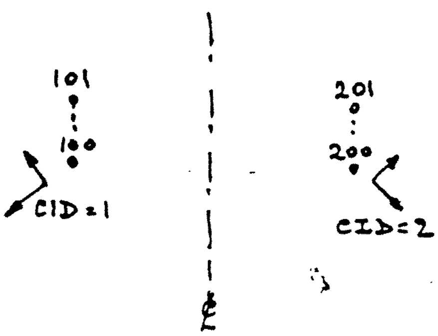




.....RBAR ID 100 101 123456 (101 DEPENDENT)

NEW GRID 101 ADDED WITH GLOBAL D.O.F. IN BASIC.

DISPLACEMENTS OF 101 CAN BE COMPARED WITH MAIN STRUCTURE



PAIR OF GRIDS 100 AND 200 HAVE TO HAVE IMPOSED ON THEM AVERAGE MOTIONS.

- CREATE NEW GRIDS 101, 201 IN BASIC D.O.F.
- LINK WITH RBARS

  RBAR ID 101 100 123456 (101 INDEPENDENT)

  RBAR ID 201 200 123456 (201 ")

- WRITE MPC'S REFERENCING NEW POINTS

  MPC ID 101 1 1.0 201 1 -1.0

  ⋮

Note on the use of dependant and independant freedoms

As mentioned previously the use of rigid elements always requires the specification or implication of dependant and independant D.O.F.

All dependant D.O.F. are put in the M set and must be exclusive. A fatal error will occur if this is not so. Thus ensure that these are referenced only once within all rigid data and constraints.

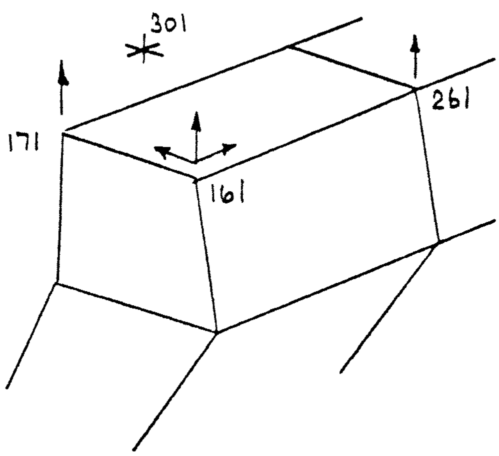
The indpendant D.O.F. are used in general to calculate the stiffness of the dependant points and usually comprise of a set of statically determinate freedoms. An error will result if any of of these freedoms are singular and alternative freedoms are not to be found in the element... an RBE1 with freedoms swapped.

Obscure errors will result if singular freedoms are referenced in RBE3 elements.

Freedoms in the S set (auto SPC set) will be removed from the set as Nastran thinks stiffness is to be obtained within the element, and thus a singular freedom may be used unwittingly in error.

Exchanging freedoms in RBE1 elements

If a dependant grid in a RBE1 equation is required to be on the boundary of a superelement, for example, a fatal error will occur if the point is left dependant. However the boundary point may be exchanged with one of the independant points and the solution will be unaffected.



RBE1 to give point 301 stiffness in freedoms 1,2,3.

```
RBE1   ID   161   123   171   13   261   3
+      UM   301   123
```

RBE1 with freedoms reversed

```
RBE1   ID   301   123   171   13   261   3
+      UM   161   123
```

CONSTRAINTS AND SUPPORTS

These two items are interlinked, since what would be regarded as a simple set of constraints to remove rigid body motion on a componant would also generate support loads if the loading was unbalanced. The elements used for constraints and supports are SPCs and MPCs, single and multi-point constraints. Other rigid body elements are used in certain applications.

The only forces output simply are SPC forces using the case control deck SPC=i and SPCFORCE=ALL statements. Other implied forces from MPCs or RBEs etc. are only seen from the GPFB output as out of balance forces at the grids involved.

Several uses are listed below.

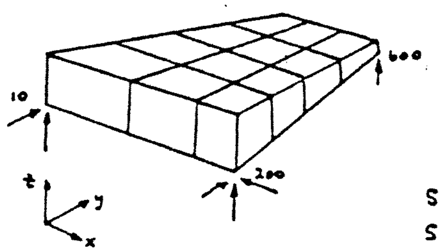
a) Statically determinate supports.
b) Boundary constraints.
c) Imposition of deflected shapes.
d) Imposition of plane sections.
e) Constraint equations using MPCs.

a) Statically determinate supports.

The minimum requirement for all analysis models is that they are supported against rigid body motion, ie. statically determinate supports or constraints. If any loading is applied to the model the reactions, SPC forces, at these points will be generated and can be checked by statics. If the loading is in balance the SPC forces will be zero or insignificant, in which case the choice of the constraint points will have no effect on the internal load distribution, and will simply serve as a base from which the displacements are calculated.

a) STATICALLY DETERMINATE SUPPORTS

THIS IS THE MINIMUM REQUIREMENT FOR ANY MODEL



SUPPORTS OR CONSTRAINTS MUST BE CAPABLE OF RESTRAINING 6 RIGID MOVEMENTS. X, Y, Z, θ_x, θ_y, θ_z.

SPC1 ID 1 200
SPC1 ID 2 10 200
SPC1 ID 3 10 200 600

SPC FORCES WILL BE RECOVERED IF LOADING IS NOT IN BALANCE.

b) Boundary constraints.

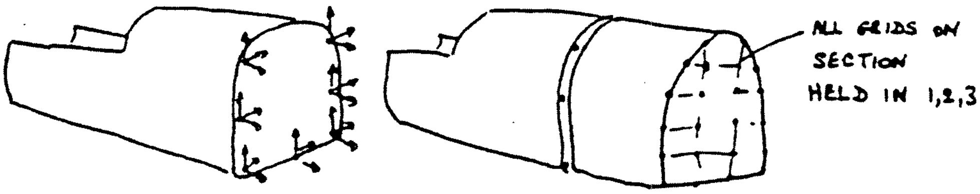
Usually the support or constraint system is redundant. A front fuselage model, for example, may be held in freedoms 1-3 at the transport joint bolt positions, or mounted on a plug of centre fuselage structure and all grids at the end of the plug held in freedoms 1-6.

A wing may be constrained at the Fuselage P/U lug points in the freedoms capable of taking load.

Similarly a test specimen model may be regarded as fully built in at a section away from the area of interest.

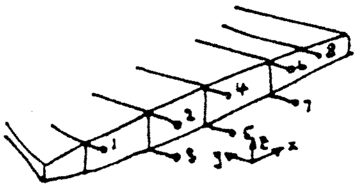
b) BOUNDARY CONSTRAINTS

MODEL IS HELD AT JOINT POSITIONS OR AT ALL GRIDS AT A SECTION



JOINT GRIDS SPC'D IN 1,2,3

SPC1 ID 123 100 110 120 121 130 ...



SPC1 ID 3 1 8 ... FRONT/REAR SHEAR

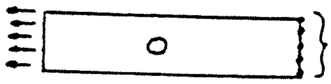
SPC1 ID 23 2 3 5 6 7

SHEAR AND BENDING

SPC1 ID 123 4 SHEAR, BENDING + DRAG

WING ROOT CONSTRAINTS

b) CONTINUED



THESE POINTS FULLY CONSTRAINED

TEST SPECIMEN

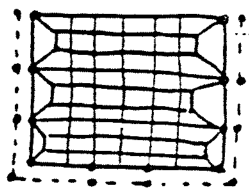
c) Imposition of deflected shapes.

The SPC card has the provision for imposing a set of displacements at any grid points and in any of the global D.O.F. One could use this facility to impose a boundary shape onto a detailed model of a component, the shape being determined from a previous analysis. This, though, is a dangerous technique, and it is better to impose boundary LOADS.

A more common use is in the analysis of pipework systems. The pipework model has imposed upon it the deflections at the attachment points obtained from a maximum deflection case for the structure to which it is attached. For example a wing fuel pipe model would have maximum wing deflections imposed at the attachment points.

The SPCD card enables enforced displacements to be requested at the Subccase level along with other loading if present. Thus several sets of deflections can be requested.

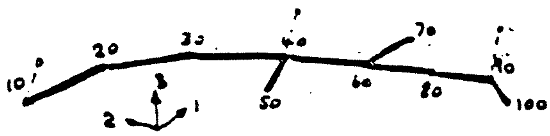
c) <u>IMPOSITION OF DEFLECTED SHAPES</u>



BOUNDARY DISPLACEMENTS IMPOSED FOR LOADING CASES.

THIS IS <u>NOT</u> RECOMMENDED.
USE BOUNDARY <u>LOADS</u>.

<u>FINE MESH PATCH</u>



DISPLACEMENTS FROM STRUCTURAL-ANALYSIS WORST CASE ARE IMPOSED ON PIPE ATTACHMENT POINTS

<u>PIPEWORK MODEL</u>

```
SPC  ID  10  3  8.0   10  1  2.0
SPC  ID  40  2  25.0  40  1  8.0   ETC
SPC  ID  90  3  23.0  90  1  12.0
```

ALTERNATIVELY SPCD CARDS MAY BE USED AND THE DEFLECTED SHAPES IMPOSED AT THE SUBCASE LEVEL

EG:-

```
SUBCASE 1
LOAD = 100
SUBCASE 2
LOAD = 200
  :
SPCD  100  10  3  8.0
SPCD  100  10  1  2.0
        :
SPCD  200  10  3  12.5
        :
```

DISPLACEMENTS SET UP AS LOAD CASES.

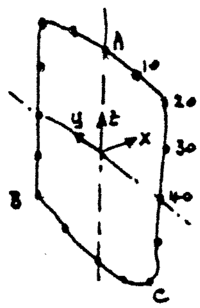
d) Imposition of plane sections.

This sort of requirement occurs when portions of a structure are modelled in isolation, and E.B.T. conditions are considered to apply at the boundaries. This implies that sections remain PLANE, and the condition can be obtained by using sets of RBE1s relating the section grid displacements to three reference grids. For 2-D models this approach will give a linear deformed shape in the plane of the model.

Since the technique FORCES a plane section, a set of self balancing forces will be generated at the grids, and these can be seen from the GPFB at the section.

d) IMPOSITION OF PLANE SECTIONS

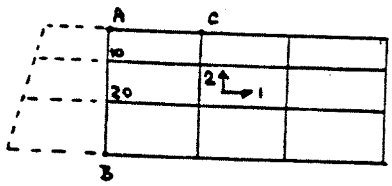
PLANE SECTIONS ARE IMPOSED TO REPRESENT E.B.T. CONDITIONS.



RBE1 EQUATIONS LINK THE OUT-OF-PLANE FREEDOMS OF GRIDS TO THREE REFERENCE POINTS A, B AND C.

```
RBE1  ID   A   123   B   12   C   1
+          UM  10    1   20   1   30  1
+              40    1   ....     ETC
+              ..    -   .        ..
```

FRAME SECTION WILL MOVE CO-PLANAR, NOT NECESSARILY PARALLEL TO THE YZ PLANE.




```
RBE1  ID   A   123   B   13   C   3
+          UM  10    1   20   1
```

2-D MODEL, END AB FORCED TO LINEAR SECTION.

e) Multipoint constraints using MPCs.

MPCs specify the behaviour of a group of points by solving a set of equations relating the DISPLACEMENTS of the points to each other. Normally these equations MUST be in balance, otherwise the model will be forced out of balance and the resulting loading will be affected.

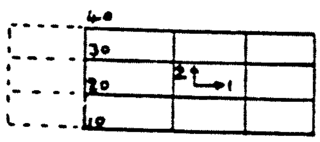
The general equation for an MPC is as follows :-

Uxa + Uxb + Uxc ... etc. = 0.0   ie. the sum of all the displacements = 0.0

An example of their use is to force the movement of a set of grids to be equal. In a simple 4 grid section the grids are to be forced to move with the same X displacement. The outer two grids can be forced to have identical displacements, Uxa = Uxb, and the inner two grids can have linear equations written to link the displacements to the outer two.

MPCs can also be used to transfer loads in specified directions between co-incident grids, at hinges etc. Again the equations would be simply Uxa = Uxb, or Uxa - Uxb = 0.0.

e) MULTIPOINT CONSTRAINTS USING MPC's



FORCING EQUAL DISPLACEMENTS ONTO SETS OF GRIDS.

$\Delta x_{10} = \Delta x_{40}$ OR $\Delta x_{10} - \Delta x_{40} = 0{\cdot}0$

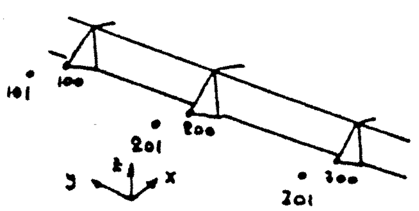
MPC ID 10 1 1·0 40 1 -1·0

POINTS 20 AND 30 CAN BE 'BEAMED' USING MPC's

MPC ID 20 1 1·0 10 1 -·6667
+ 40 1 -·3333
MPC ID 30 1 1·0 10 1 -·3333
+ 40 1 -·6667

e) CONTINUED

TRANSFER OF LOADS BETWEEN STRUCTURES - MPC's



PAIRS OF COINCIDENT GRIDS 100,101 ETC.

100,100 TO CARRY X & Z LOADS
200,201 " " Z "
300,301 " " X,Y,Z "

MPC EQUATIONS WOULD BE :-
MPC ID 100 1 1·0 101 1 -1·0
MPC ID 100 3 1·0 101 3 -1·0
MPC ID 200 3 1·0 201 3 -1·0
MPC ID 300 1 1·0 301 1 -1·0
MPC ID 300 2 1·0 301 2 -1·0
MPC ID 300 3 1·0 301 3 -1·0

ERRORS IN NASTRAN ELEMENTS

Nastran is a displacement method finite element program and as such derives all its information from the solution of the stiffness matrix and the displacements of the grid points. Thus the behaviour stiffnes wise of each element under a variety of loading conditions should be as accurate as possible. Elements which have an incorrect stiffness will degrade the solution and thus the final stress distribution in the structure.

Recent developments in structural optimisation using aeroelastic criteria have brought to light the need to predict deflected shapes accurately, for critical items such as flaps in particular.

The derivation of the stiffness of any of the elements is simple provided the elements are regular and un-warped, ie. rectangular and planar. Unfortunately Nastran has to deal with all conceivable geometry variations and in general the more obscure the shape the more difficult it is to arrive at an acceptable stiffness matrix. All elements are accurate for pure tension/compression loads regardless of their shape.

A Nastran application note of March 1984 deals with the problem of element accuracy in some detail and highlights particular situations where large errors are to be expected. The method of detecting the errors is to subdivide a regular structure, for which the theoretical stiffness is known, into regular and irregular elements. The comparison of the displacements for simple loading cases then gives an indication of the element stiffness accuracy for the various shapes and for various element types.

The results for a cantilever beam are shown on the following page. For rectangular element geometry all elements (the QUAD2 is now obsolete) behave well under all loading conditions. For trapezoidal and parallelogram elements some shear terms are as low as 5% of the theoretical values. The QUAD4 is poor only for in plane shear whilst the HEXA is poor for both in plane and out of plane shear. Increasing the number of elements through the depth would improve the accuracy as expected.

For a rectangular plate subject to uniform or point loads the results are shown on the second sheet. The centre deflection errors can be seen to be small in all cases with the exception of the HEX20 being poor in most cases, and the QUAD4 being poor for low mesh size and high aspect ratio elements.

The conclusions reached in the note are that whilst all elements produced failures due to locking or mechanisms in certain instances and none met all the tests, the general set of elements if used with these failures in mind can be used with confidence.

At Warton a series of single element tests have over a period of time produced similar conclusions, and evidence from actual model comparisons have led to the following recommendations.

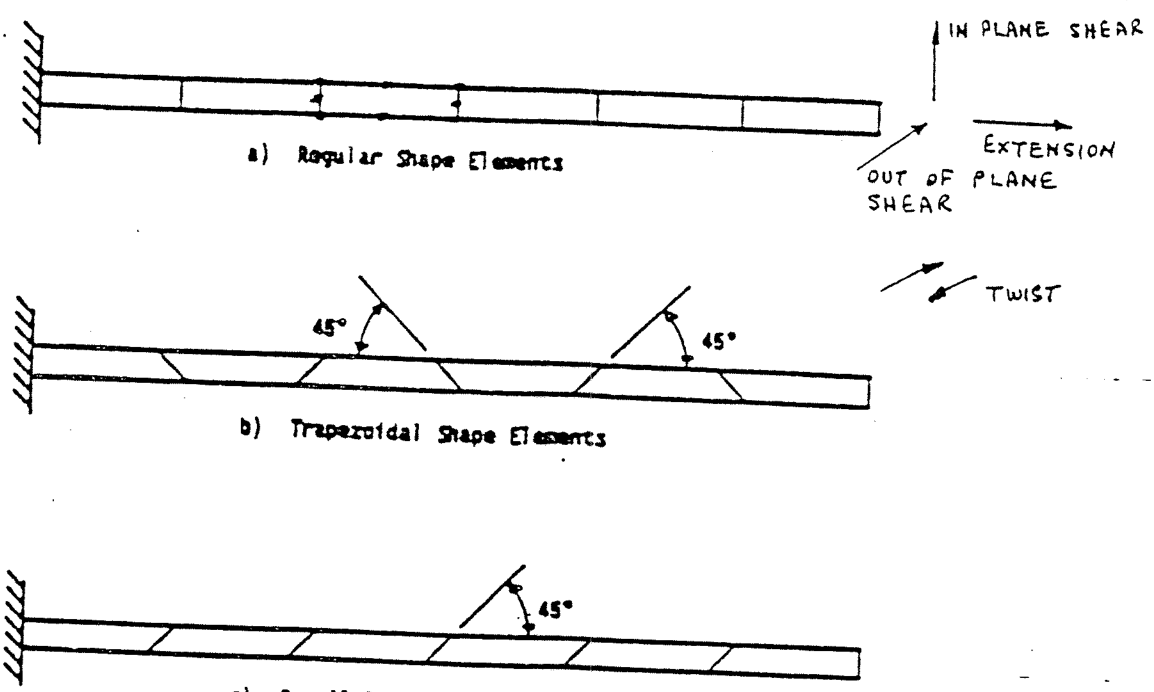
a) Avoid obscure element shapes where high loads are input .. use low aspect ratio rectangular elements. It is the high strain gradients which cause the stiffness problems ... uniform end loads are dealt with correctly.

b) For deflection critical items such as flaps, use a mesh size of at least 4 across the chord. A mesh size of 2 will give rise to an over stiffness of 100% in torsion.

c) Refine the mesh at high load input points and coarsen up at least two elements away from the area.

d) Avoid using non uniform solid elements where out of plane displacements are critical ... use QUAD4 bending elements.

STRAIGHT CANTILEVER BEAM TESTS



a) Regular Shape Elements

b) Trapezoidal Shape Elements

c) Parallelogram Shape Elements

Table 8. Results for Straight Cantilever Beam

Tip Loading Direction	Normalized Tip Displacement in Direction of Load					
	QUAD2	QUAD4	QUAD8	HEXA(8)	HEX20	HEX20(R)*
(a) Rectangular Elements						
Extension	.992	.995	.999	.988	.994	.999
In-Plane Shear	.032	.904	.987	.981	.970	.984
Out-of-Plane Shear	.971	.986	.991	.981	.961	.972
Twist	.566	.941	.950	.910	.904	.911
(b) Trapezoidal Elements						
Extension	.992	.996	.999	.989	.994	.999
In-Plane Shear	.016	.071	.946	.069	.886	.964
Out-of-Plane Shear	.963	.968	.998	.051	.920	.964
Twist	.616	.951	.943	.906	.904	.918
(c) Parallelogram Elements						
Extension	.992	.996	.999	.989	.994	.999
In-Plane Shear	.014	.080	.995	.080	.967	.994
Out-of-Plane Shear	.961	.977	.985	.055	.941	.961
Twist	.615	.945	.965	.910	.904	.913

*The good to excellent results shown here for HEX20(R) were obtained in spite of singularity in the beam's stiffness matrix.

RECTANGULAR PLATE TEST

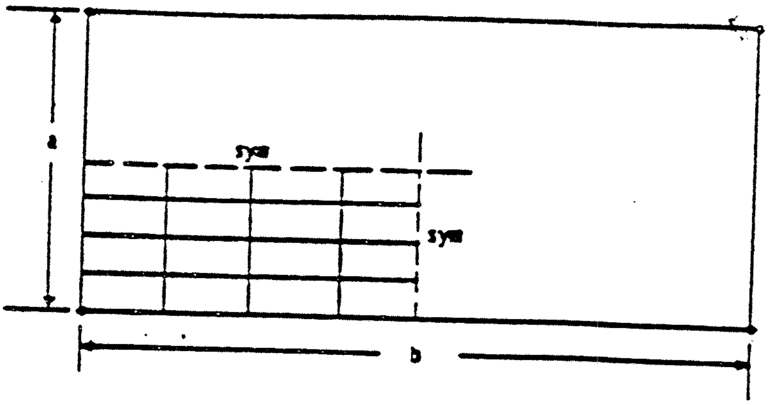


Table 11. Results for Rectangular Plate Simple Supports; Uniform Load.

(a) Aspect Ratio = 1.0

Number of Node Spaces* Per Edge of Model	QUAD2	QUAD4	QUAD8	HEXA(8)	HEX20	HEX20(R)
	Normalized Lateral Deflection at Center					
2	.971	.981	.927	.989	.023	1.073
4	.995	1.004	.996	.998	.738	.993
6	.998	1.003	.999	.999	.967	1.011
8	.999	1.002	1.000	1.000	.991	1.008

(b) Aspect Ratio = 5.0

Number of Node Spaces* Per Edge of Model	QUAD2	QUAD4	QUAD8	HEXA(8)	HEX20	HEX20(R)
	Normalized Lateral Deflection at Center					
2	.773	1.052	1.223	.955	.028	1.139
4	.968	.991	1.003	.978	.693	.995
6	.993	.997	1.000	.990	1.066	1.024
8	.998	.998	1.000	.995	1.026	1.006

*For elements with midside nodes, the number of elements per edge of model is equal to one-half the number of node spaces.

Table 12. Results of Rectangular Plate Clamped Supports; Concentrated Load.

(a) Aspect Ratio = 1.0

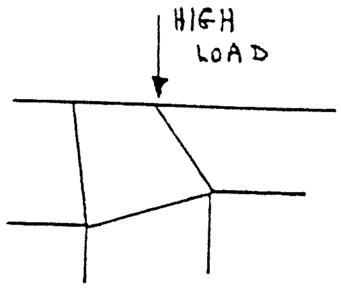
Number of Node Spaces Per Edge of Model	QUAD2	QUAD4	QUAD8	HEXA(8)	HEX20	HEX20(R)
	Normalized Lateral Deflection at Center					
2	.979	.994	1.076	.885	.002	.983
4	1.008	1.010	.969	.972	.072	.433
6	1.006	1.012	.992	.988	.552	.813
8	1.005	1.010	.997	.994	.821	.942

(b) Aspect Ratio = 5.0

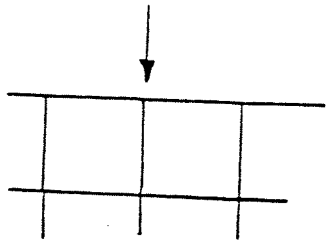
Number of Node Spaces Per Edge of Model	QUAD2	QUAD4	QUAD8	HEXA(8)	HEX20	HEX20(R)
	Normalized Lateral Deflection at Center					
2	.333	.519	.542	.321	.001	.363
4	.512	.863	.754	.850	.041	.447
6	.638	.940	.932	.927	.220	.721
8	.723	.972	.975	.957	.374	.867

METHODS OF AVOIDING ERRORS IN MODELLING

a) AVOID OBSCURE SHAPES WHERE HIGH LOADS ARE INPUT

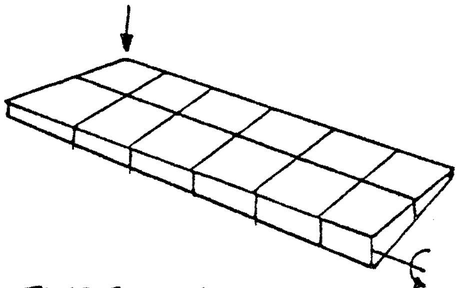


LARGE ERRORS

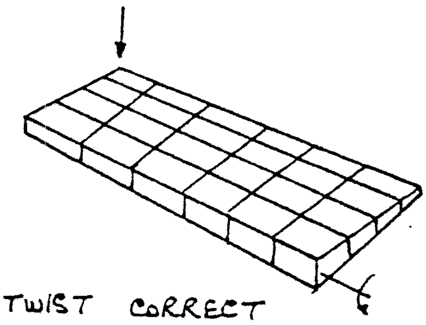


SMALL ERRORS

b) DO NOT USE COARSE MESH ON THIN ITEMS

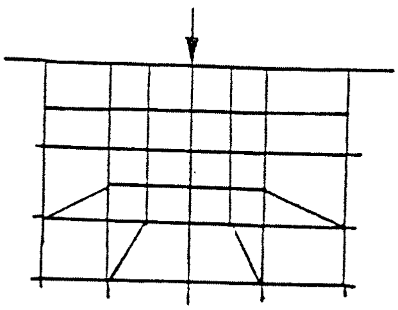


TWIST +100% STIFFNESS

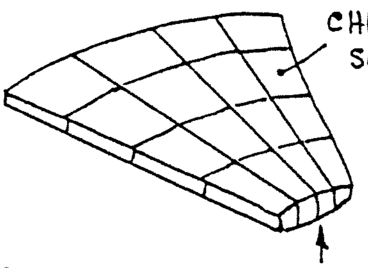


TWIST CORRECT

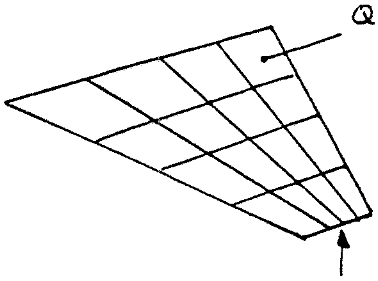
c) REFINE MESH AT HIGH LOAD POINTS



d) AVOID SOLID ELEMENTS WITH NON PARALLEL FACES FOR OUT OF PLANE CRITICAL MODELS.



COULD BE UP TO 5 TIMES TOO STIFF IN BENDING



TIP DISPLACEMENT CORRECT

APPLICATION OF LOADING

The application of the loading is an important part of structural analysis. One could say that without loading all models are unusable.

At the initiation of the model it is vital to know the type of loading to be applied, as this can affect the modelling process. A wrong assumption can render the model useless or produce meaningless results.

The loading can be broken down into several distinct types.

a) Symmetric and Antisymmetric components
b) unit cases
c) point loads
d) distributed loads
e) pressure loads
f) inertia loads
g) fully balanced cases

Each of the types can load the model using any of the available Nastran loading cards. The most commonly used being FORCE, MOMENT and PLOAD cards. Methods of combining the types of loading include the use of the LOAD card, and in the case controll deck the subcases can be combined in any linear manner.

A description of the common cards used follows.

1. FORCE ID G CID F N1 N2 N3

The FORCE card applies direct forces at grid point G, of magnitude F and in the direction given by the vector N1,N2,N3 relative to the coordinate system CID. For example a force of 2.8Kn applied in the Z direction of the basic system to grid 200 can be defined by

```
FORCE   ID   200        2800.   0.0   0.0   1.0
or
FORCE   ID   200        1000.   0.0   0.0   2.8
```

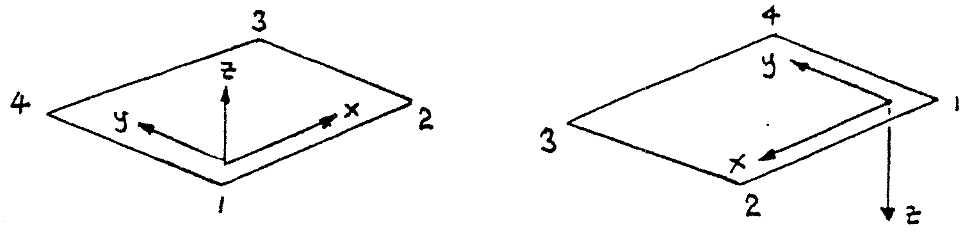
2. MOMENT ID G CID M N1 N2 N3

The MOMENT card applies a moment load to grid point G, of magnitude M and in the direction given by vector N1,N2,N3 relative to the coordinate system CID. A moment of 5.0KnM appied to grid 200 in the MX direction (basic) ...

```
MOMENT  ID   200        5.0+6      1.0   0.0   0.0
or
MOMENT  ID   200        1000000.   5.0   0.0   0.0
```

3. PLOAD ID P G1 G2 G3 G4

The PLOAD card applies a pressure P to the panel defined by grids G1 to G4, or G1 to G3 in the case of triangular panels. The pressure loading results in direct loads at the corner grids, the DIRECTION of which is related to the ORDER of the grids on the card. X is in direction G1 to G2, Y into the panel, and Z as defined by the right hand rule. The pressure loading acts as forces in the Z direction. Thus defining the grids in different orders results in different POSITIVE pressure directions, but this can be corrected by changing the sign of the pressures on the offending panels.



+VE PRESSURE GIVES LOAD IN THE ELEMENT LOCAL Z DIRECTION.

4. PLOAD2 ID P E1 E2 E3 E4 ...etc

This is a simpler pressure loading card in that a constant pressure P is applied to panel elements E1,E2,E3 etc. POSITIVE pressure gives rise to loads in the element local axis Z direction, see above, and care should be taken to check that the loading is as expected.

5. PLOAD4 ID EID P1 P2 P3 P4 ... ..

This pressure loading card is used if the pressures are varying across the panel in that pressures at the corners of the element can be specified. P1 refers to G1, P2 to G2 etc. where G1 - G4 are the grids defining the panel connections. Fot triangular panels only P1 to P3 are defined. The positive direction of the loading applies as above.

6. LOAD ID S S1 ID1 S2 ID2 S3 ID3

The LOAD card combines any of the previously set up types or componants of loading to form a single load case for use in the requested analysis run. Load case ID is formed by adding together the case ID1 multiplied by the scalar S1, ID2*S2, ID3*S3 etc, with an overall scalar of S. Thus in the following example :-

Aerodynamic load ID	= 10
Inertia	= 20
U/C	= 30

The aero loads have to have a scalar of 2.0, the inertia a scalar of -9.0, and the U/C loads a scalar of 1.0, with an overall ultimate factor of 1.5. The resulting LOAD card is :-

LOAD ID 1.5 2.0 10 -9.0 20 1.0 30

DEFINITION OF THE LOADING

All loading, other than simple cases to check the model, should have an adequate description/definition supplied with the data. Items should include :-

1. Name .. case identifier etc
2. Configuration .. flight parameters, c.g. position, mass, with/without stores + details, fuel state etc.
3. Type of loading .. aerodynamic, inertia, net etc.
4. Sign Convention
5. Condition .. Proof/Ultimate (UF=..)
6. Data set name and where to find the data, if on a file
7. Loading details if not on file
8. Balance information .. totals about a reference point

There is a general requirement, on EFA, for all loading to be supplied in a consistant manner, where the sign conventions and balance information are capable of a simple transformation to the basic A/C axes. ie. X aft, Y to stbd, Z up, and the rotations following the right hand rule. All accelerations also follow this sign convention. No left hand sign conventions should be used.

However this has yet to be implimented fully and a mixture of sign conventions, even for the same componant, appear in the loading documentation. Thus be carefull to check the sign conventions used, if they are available, and convert these to the basic A/C system (or your model system) for checking.

The balance information is vital. It is one thing to apply a pressure distribution or set of loads, it another thing to be in a position to check that the loading is correct. If the summation of the loading about a reference point is known from the loading document then checking is easy, if it is not known then ASK for it to be supplied.

CHECKING THE LOADING

The Nastran runs will output the overall applied loading resultants, but it is important to check each componant of the loading before it is used in the analysis.

The PPS check loading facility should be used for this, as it can be run online on the Vax. Both overall resultants and shear, moment and torque about an axis (which may be swept)can be generated. It is quicker to use PPS than wait for Nastran runs on the mainframe.

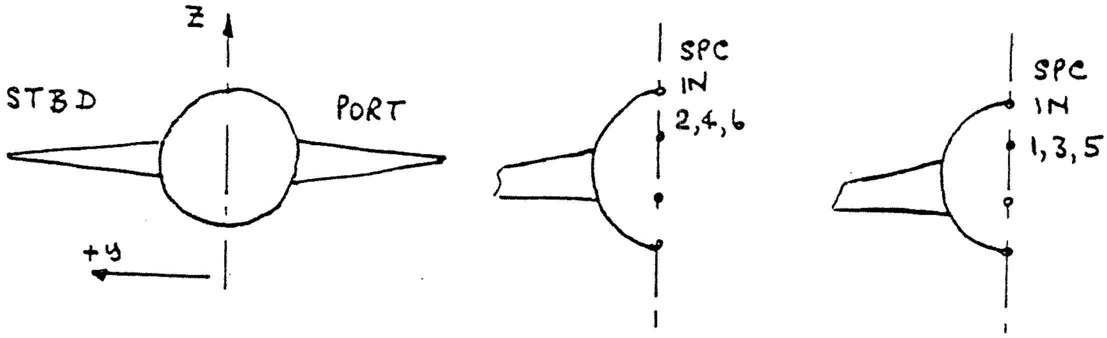
All loading should be checked against the documentation before proceeding with the analysis runs.

a) Symmetric and Antisymmetric components

Loading applied in particular to fuselages and wings can normally be sub-divided into symmetric and antisymmetric components. This is vital if only a half fuselage has been modelled. The two components of loading are thus applied to the half model with the model constrained in the relevent manner. Thus symmetric loading will be applied to the model constrained symmetrically, and antisymmetric loading applied to the model constrained antisymmetrically.

The two resulting sets of subcases can be combined to give the correct Port or Starboard output, which will be Asymmetric.

The method of splitting up the loading is shown below.

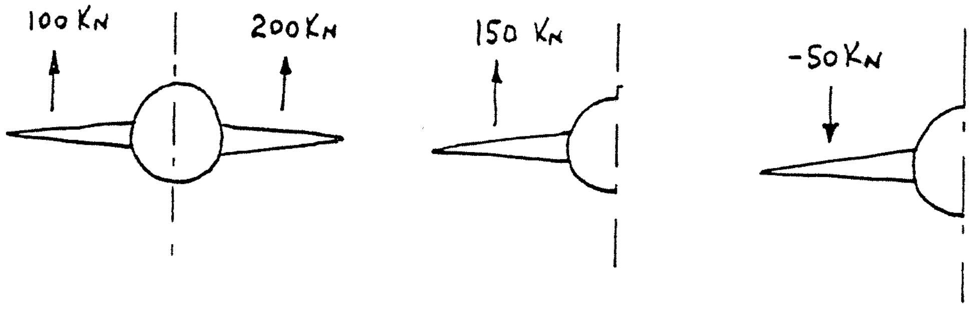


Full model Symmetric Antisymmetric

The full model loading can be split up into the components using the folowing formulae. It is assumed tha Starboard is +ve Y.

Symmetric component = 1/2 Sum (Stbd + Port)

Antisymmetric component = 1/2 Difference (Stbd - Port)



Total loading Symmetric Antisymmetric

The true Stbd loading is thus Symm+A/S = 150-50 = 100 Kn
The true Port loading is thus Symm-A/S = 150+50 = 200 Kn

This process can be carried out on any fuselage/wing assembly, and on any structure modelled with planes of symmetry.

b) Unit cases

By definition these are cases set up in order to be used either alone for checking purposes or by scalar combinations to form up actual cases. Types of unit cases are :-

1. Checking cases .. unit shear, moment, torque etc., unit loads on tips of wings, flaps etc.

2. Unit actual cases .. 1g inertia, 1.0N/mm^2 internal pressure, store attachment cases, U/C cases etc.

3. Balancing cases .. unit cases consisting of distributed (perhaps coarse) loads with unit resultants about the origin. These can be used to trim existing cases to meet the exact balance resultants. This is usefull for full A/C work or in setting up non-critical cases where the resultants only are known.

4. Unit cases for flexibilities etc. .. Flexibilities are normally produced by inverting the KAA matrix. However if this is not possible unit load cases have to be set up for each freedom required and the resulting displacements at the points used.

5. Rigid body movement cases .. The normal checks of applying unit cases to a structure should also include rigid body movement cases where the structure has imposed upon it displacements corresponding to the 6 D.O.F. present.

   The structure is supported in a statically determinate manner and the 6 rigid body motions applied using SPCD data applied to the support D.O.F. Thus there will be 6 subcases, typical translations being 100mm and rotations 0.1 Radians.

   There should be no significant SPC forces recovered, ie. the structure should be ~~stress free for rigid body motion.~~

   These checks are of particular importance if models are to be passed outside the department in that they represent a more severe test than is usually carried out.

   The reasons for the use of these cases are as follows.

   a) insignificant SPC forcces produced from a loading case may occur at areas of low displacement. Rigid body checks may show these to be significant and thus some fix must be applied to the model.

   b) if the model is part of a large assembly of super-elements, checks on local rigid supports may not reveal significant SPC forces. However the portion of structure may be subject to large overall movements when assembled (a fin for example moves bodily on the rear fuselage) and these small SPC forces may be significant.

   c) free-free stiffness matrices (KAA's) should have rigid body checks carried out on them if they are to be passed outside the department or issued officially and archived.

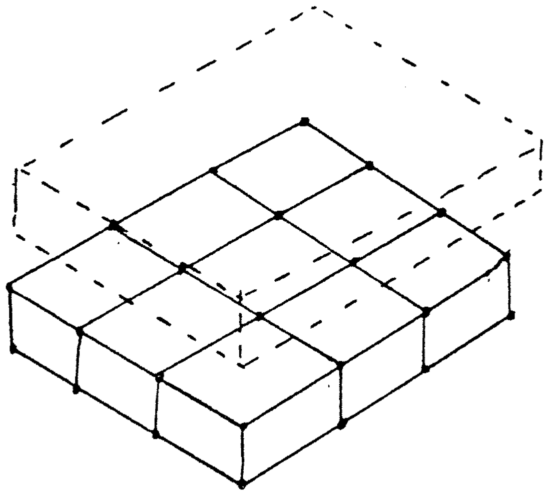
d) similarly free-free matrices recieved should always be checked for rigid body behaviour before use.

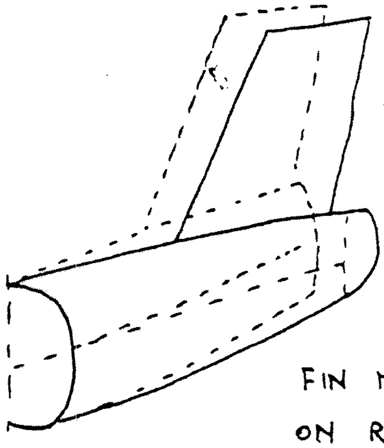
The Nastran deck for carrying out the checks should contain the following items.

1. SPC data to hold the structure satically determinate and referenced above the subcase level.
2. 6 subcases calling up enforced displacements using LOAD=n, where n = 1 to 6.
3. SPCD data setting up the 6 cases using the D.O.F. given in 1. and having ID's 1-6.
4. dummy load data for each subcase, ID = 1 to 6.

The run should request all displacements and SPC forces. The displacements for each case should have visible the enforced case and the SPC forces should be insignificant.

The following page illustrates the method.

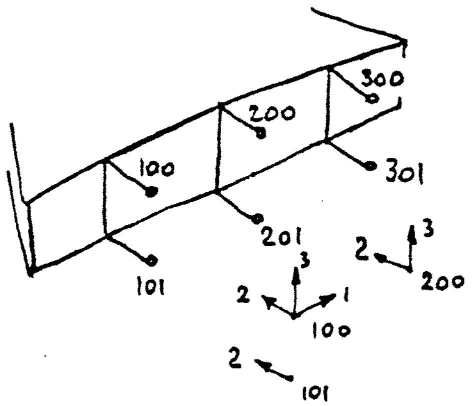




FIN MOVES BODILY ON REAR FUSELAGE.

STRUCTURE WITH RIGID BODY MOVEMENT SHOULD RECOVER NO INTERNAL STRESSES OR SPC FORCES.

RIGID BODY MOTION CHECKS



STATIC SUPPORTS

SPC1	ID	1	100		
SPC1	ID	2	100	101	200
SPC1	ID	3	100	200	

SPCD DATA

SPCD	1	100	1	100.0	... 100 mm X
SPCD	2	100	2	100.0	100 mm Y
SPCD	2	101	2	100.0	
SPCD	2	200	2	100.0	
:					
SPCD	4	100	2	20.0	0.1 Rad M_x
SPCD	4	200	2	20.0	
SPCD	4	101	2	-20.0	
:					
SPCD	6				

DUMMY FORCE DATA

FORCE 1 100 0.0 1.0 0.0 0.0
2
:

TYPICAL NASTRAN DECK

```
SPC = ID
DISPLACEMENT = ALL
SPCFORCE = ALL
SUBCASE 1
LOAD = 1                      ...... APPLIES SPCD = 1 DISPLACEMENTS
LABEL = 100 mm X MOVEMENT
SUBCASE 2
LOAD = 2                      ...... APPLIES SPCD = 2 DISPLACEMENTS
LABEL = 100 mm Y MOVEMENT
```

c) Point loads.

These can be from many sources :-

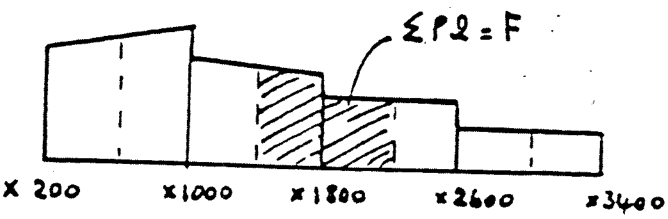
1. local calculated loading .. due to equipment inertia not modelled, U/C or engine loads, from attached items .. foreplane loading can be applied directly to a reference point if it is connected via a RBE1 to the bearing and jack points.

2. Calculated simple loading .. test specimens etc may have uniform loading which can be easily calculated using the geometry of the grids on the section.

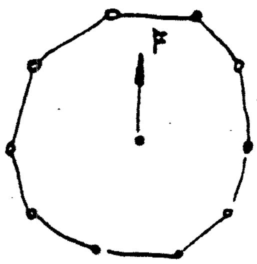
3. Applied directly .. from the loading given in the loading documents.

d) Distributed loading

Often the aerodynamic and inertia loading is seperated and supplied as shear, moment and torque diagrams along a fuselage, for example. The analysis user must spread this loading onto the available grids in whatever detail is required. For fuselages this can be carried out by dividing the diagrams into blocks and applying the block loading to grids at the available frame stations. The frame loading can thus be as detailed as necessary, applying the resultant to all the grids or to a generalised point linked to a selected set of 'hard' points. The final distributed loading must be checked against the known resultants and trimmed if not correct.



AERODYNAMIC SHEAR LOADING ALONG FUSELAGE



FRAME X 1800

LOADING F APPLIED TO ALL FRAME GRIDS OR APPLIED TO GENERALISED POINT ONLY

e) Pressure loads

Pressure loading has to be applied to a model in the following instances :-

1. Flying surface pressures which generate lift.
2. Internal pressures .. cabin, intake, fuel etc
3. Extrenal pressures .. canopy side loads, radome loads etc.
4. Blanket pressures for designing local structure.

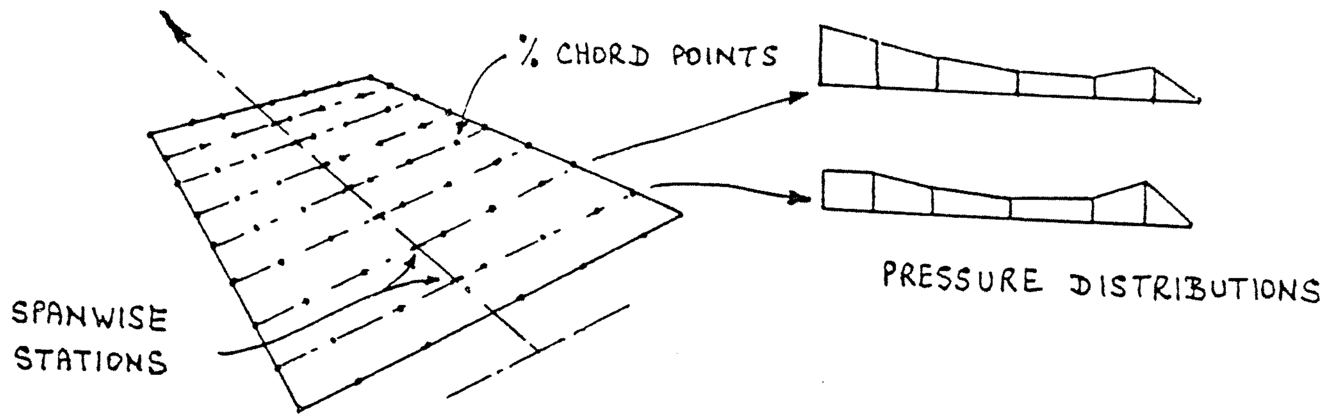
The pressure loading must be applied by hand using PLOAD data, or by using the PPS pressure loading facilities ..for pressure loading and for internal pressure generation due to fuel inertia.

Application by hand is frought with the problems of panel local axes, and should be applied in stages, one face at at time until a balance is obtained. If in doubt use the Case Control card OLOAD=ALL. This will print out the forces on the grids from the PLOAD data and any sign reversal should easily be seen. It may be usefull to run a constant pressure case for checking out.

The PPS pressure loading faclity.

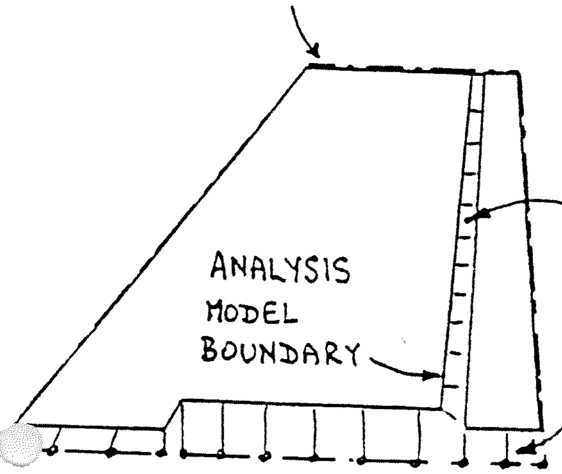
Using the PPS pressure loading facility, the basic data is normally obtained from a file on the mainframe. This consists of a set of spanwise locations and chordwise percentages at which pressures are defined, followed by a table of pressures. PPS applies these pressures to a set of surface elements and generates either a set of PLOAD cards (taking into account the loacl axes) or a set of FORCE cards for the case.
The user should set up a set of surface panels for the loading and fill in any gaps between controll surfaces etc. with dummy elements. If the analysis planform does not match or cover the loading document planform, either the panels should be adjusted in shape or additional grids and panels added to give EXACTLY the document planform. Refer back to the document source if excessive discrepencies are present.
Applying the pressure cases should then give the required resultants as given in the loading doumemts.
Adjustments made to the loading planform can then be either ammended back to the analysis condition and the loading re-generated, if acceptable, or the loads on the additional grids disarded. This adjustment in loading should be noted since for balanced cases the anlysis loading may now be incorrect.

PRESSURE LOADING DATA - ON WINGS ETC.



- PRESSURES ARE DEFINED FOR EVERY % CHORD POINT AT EVERY SPANWISE STATION.

LOADING PLANFORM



- FOR CHECKING PURPOSES LOADING PANELS SHOULD BE MADE UP TO LOADING DOCUMENT PLANFORM.

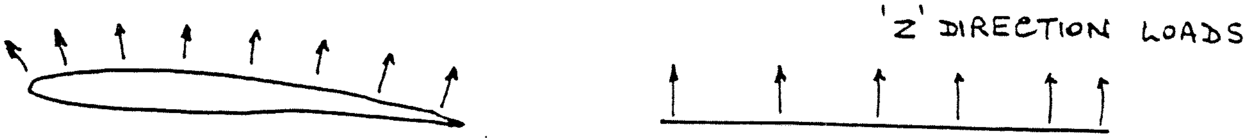
FILL IN ANY GAPS IN MODEL.

ADD EXTRA GRIDS AND PANELS TO MAKE UP PLANFORM.

- CHECK LOADING USING PPS - SHOULD TIE UP WITH DOCUMENTATION.
- DISCARD LOADING ON EXTRA GRIDS - LOSS IN LOAD MAY BE OF INTEREST IN BALANCED CASES.

OUT OF PLANE GEOMETRY

- LOADING DOCUMENTATION ASSUMES A FLAT-PLATE SURFACE.
- CORRECT LOADING WILL BE GENERATED SETTING Z = 0.0 FOR THE PANEL GEOMETRY.



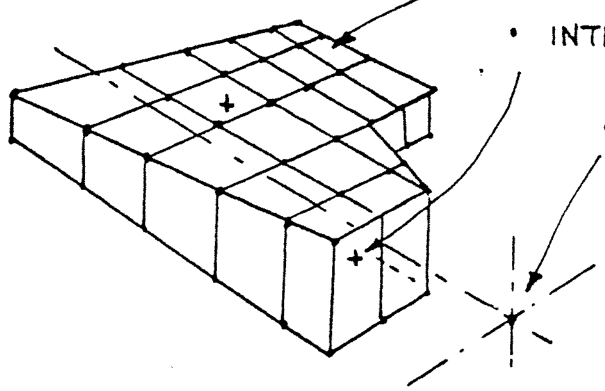
ACTUAL WING SHAPE AND DIRECTION OF LOADING

LOADING DOCUMENT ASSUMES FLAT PLATE

Geometry of Loading Panels ... The aerodynamic theory which produces the flying surface loading assumes that the surface is a flat plate, and that the loading is normal to this. In addition if the surface has a small dihedral the loading may still be specified as being in the A/C 'Z' direction. Thus the user should find out what assumptions have been made in the loading, and create the loading panel geometry likewise, at a constant Z or on a flat plane using a local coordinate system.
Bt implication drag or thrust loading due to the curvature of the real surface is not generated from the pressure distributions, these effects must be added later.

Fuel Pressures ..
The PPS Fuel Tank Pressure program calculates the internal fuel pressures and inertia effects of a fuel tank subjected to any form of linear/angular accelerations and angular velocities. The tank is defined by a set of panel elements and geometry on the boundary together with additional internal points to indicate which side of the panels are internal to the tank. The centre of rotation, fuel density and system pressure are also input together with the controll data to define the inertia cases required.
The loading may be output as FORCE, PLOAD2 or OPTPRES cards, the latter being used for optimisation purposes. The load summations are printed out together with the fuel inertia properties relative to the tank cg. Since in general the model will not correspond to the real tank due to internal structure volumes not being accounted for, a unit pressure case should be run in order to find the volume and mass. If the mass is different to the required mass the fuel density should be adjusted before actual cases are run.
The program is easey to use and has been used extensively on the EFA wing to produce fuel tank loading.

FUEL PRESSURE DATA



- PANELS AND GEOMETRY
- INTERNAL DEFINITION POINTS
- CENTRE OF ROTATION
- FUEL DESITY
- SYSTEM PRESSURE

- CASES DEFINED BY LINEAR/ANGULAR ACCELERATIONS AND VELOCITIES
- OUTPUT CAN BE FORCE, PLOAD2 OR OPTPRES DATA
- TANK INERTIA CALCULATED
- LOAD SUMMATIONS OUTPUT

FACTOR DENSITY TO ACCOUNT FOR REDUCED VOLUME (HENCE MASS) OF FUEL DUE TO INTERNAL STRUCTURE

f) INERTIA LOADING

Inertia loading forms a vital part of many loading cases. Several types may be encountered, structural, fuel, concentrated (non structural) mass etc.

1. Structural inertia loading can be supplied in many ways. Fuselages tend to be defined by running mass diagrams, Wings etc. by mass distributions, some times by mass per unit area in preliminary calculations.
   However supplied, the inertia cases must be formed from the mass definition.

   Recently the trend has been to define the mass distribution directly at a set of interface grids. The weights department carry out this task, and their data is converted to CONM2 card data giving the mass at the grids. In fuselages the mass of two half bays may be lumped to a generalised point at the associated frame position. In this case the CONM2 data will include the inertia properties of the section.

   CONM2 data can be used directly by the PPS Inertia Loading program, where the mass and geometry information is used to generate equivalent FORCE data for any combination of inertia parameters. This method has been used extensively on EFA.

2. Fuel Inertia may be generated from the fuel mass allocated to the tank panel grids, either as CONM2 data for use with the PPS inertia program (in which case no fuel PRESSURES will be generated), or as unit FORCE data for linear accelerations only. Alternatively the fuel pressure loading program in PPS may be used in which case the correct fuel inertia FORCE data will be generated together with the pressure effects.

3. Concentrated items of mass can be treated in several ways. The mass may be allocated to a set of grids and the inertia effects calcuated by hand. It may be lumped to a grid at the cg of the mass and this grid connected to the structure using rigid or elastic elements..the grid then having inertia forces calculat by hand. Large masses may have self inertia which requires aplyuing as rorational forces. If several concentrated masses are present it is better to deal with them as a whole and use the PPS Inertia Program to calculate the loading.

CHECKS

Always check that the resulting inertia loading is as expected in the loading document. A change in mass specification may give rise to discrepanccies which need to be resolved before acceptance of the inertia loading.

g) Balanced Cases

All loading cases should have balance information in order to check the case, and some slight out of balance may be acceptable. In the case of flying surface loading the planform of the model usually does not correspond exactly with the loading document, and thus a compromise loading may be accepted.

However for full A/C cases an EXACT balance is necessary. The full A/C model should be supported on a set of statically determinate freedoms at relatively 'hard' points. Small resulting SPC forces may finally be seen, but these should be insignificant compared to the local stress levels(<< 1.0Kn).

A full A/C model may consist of seperate componants assembled in a reduced form using Superelements. The agreed set of loading cases have to be applied to all components and the resultants on each componant should agree to a pre-determined total. Once this is obtained the models can then be interacted to generate the interface loads and post interaction output produced.

The organisation necessary to impliment balanced loading is extensive, the items to be agreed include :-

1. Critical loading cases

2. Mass standard

3. Componant boundaries for loading .. load sharing may have to be carried out between componants, eg.fairing loads

4. Standard of loading .. not all compnants need to be loaded in detail for all cases, eg. front fuse load cases would not require detailed Fin loading.

A matrix type of table should be drawn up to define the type of loading and responsibility for all balanced cases.

In order to ensure that cases are in balance, each case should have a full A/C balance sheet, and all componants should be trimmed to give EXACTLY the correct loads.

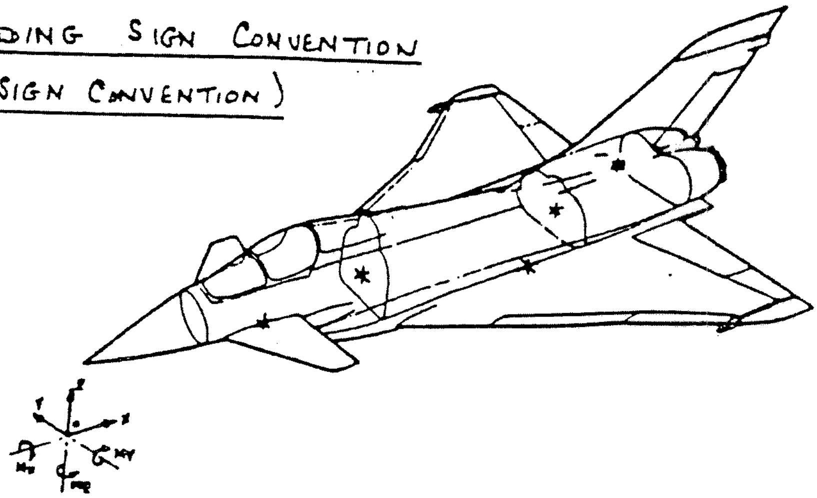
A typical type of full A/C balance sheet is shown on the next page.

LOAD CASE :

NET LIMIT LOADS

A/C LOADING SIGN CONVENTION

(BASIC SIGN CONVENTION)



LOAD REFERENCE POINTS			
COMP'T	X	Y	Z
F/FUSE			
INTAKE			
C/FUSE			
R/FUSE			
ST. WING			
ST. F/PL			
FIN			

ALL LOADS QUOTED AT REFERENCE POINTS ARE IN A/C SYSTEM SHOWN

LOADS :- KN + KNM

COMPONENT	LOADING RESULTANTS AT COMPONENT REF. POINTS					
	X	Y	Z	Mx	My	Mz
FRONT FUSELAGE						
INTAKE						
CENTRE FUSELAGE						
REAR FUSELAGE						
PORT WING						
STBD. WING						
PORT FOREPLANE						
STBD. FOREPLANE						
FIN						

	LOADING RESULTANTS AT A/C ORIGIN 'O'					
FRONT FUSELAGE						
INTAKE						
CENTRE FUSELAGE						
REAR FUSELAGE						
PORT WING						
STBD. WING						
PORT FOREPLANE						
STBD. FOREPLANE						
FIN						
TOTALS						

Inertia Parameters	$\frac{a_x}{g}$	$\frac{a_y}{g}$	$\frac{a_z}{g}$	$\dot{p}$ r/s²	$\dot{q}$ r/s²	$\dot{r}$ r/s²	p r/s	q r/s	r r/s

Flight Parameters	Mach	Alt kft	n	δ_1	δ_2	δ_3	δ_4	$\varepsilon_{\text{FULL SPAN}}$	ζ

Mass State	Mass	X_{cg}	Y_{cg}	Z_{cg}	Config