

inTENS

Integrated Program Suite for Tension Structure Design

Tensys Limited
1 St Swithins Yard
Walcot Street
Bath BA1 5BG UK

T +44 (0)1225 321950
F +44 (0)1225 321969
E mail@tensys.com

inTENS

Integrated Program Suite for Tension Structure Design

Contents

- 1 Introduction
- 2 Software Development and Testing
- 3 Numerical Modelling and Element Library
- 4 inTENS Program Modules
- 5 Output Results and Their Interpretation

Appendices

- A Tensys Limited Company Profile
- B Dynamic Relaxation
- C Data File Structures

Tensys Limited
1 St Swithins Yard
Walcot Street
Bath BA1 5BG UK

T +44 1225 321950
F +44 1225 321969
E mail@tensys.com

1. INTRODUCTION

The inTENS program suite provides the full range of computational software necessary for the design and fabrication of tension structures, specifically those utilising stressed structural membranes (typically PVC coated polyester or PTFE coated glass fabrics or ETFE foils) and cables.

The principal stages of analysis are :

1.1 Form Generation

The establishment of an equilibrium form in which specified surface element stresses are balanced by forces in the supporting structure (cable elements or beam elements or direct foundation connections to fixed points). Form generation is generally undertaken for a weightless state and the basic controls on shape are the specification of stress, force or length. Where material elastic properties are adopted it is for the purpose of imposing length controls, and the property values employed need not be realistic.

1.2 Load Analysis

The specified stress equilibrium state established at the formfinding stage may be exactly replaced by an equivalent elastically controlled model with realistic material properties. This model may then be subjected to applied loading to determine stresses and deformations under wind or snow loading. Real material selfweights are also included at this stage.

1.3 Fabrication Geometry

Tension structures are prefabricated away from site from components which need a high degree of geometric accuracy. This is best achieved if all necessary geometric data is taken from a single computer model.

In the inTENS program suite there are modules to schedule membrane cutting patterns, cable system lengths and the geometric data (plate angles etc) necessary for support structure components. Output may also be produced in formats suitable for direct use in computer aided fabrication processes.

2 SOFTWARE DEVELOPMENT AND TESTING

The inTENS program suite has been developed by the director of Tensys Ltd, Dr DS Wakefield, and is supported and maintained under his direct supervision. Details of his experience and the activities of Tensys Ltd are detailed in Appendix A : Company Profile.

The numerical analyses employed in the programs are based upon the method of Dynamic Relaxation (DR) which is described in detail in Appendix B of this document. The method was originally developed for tension structures at City University, London, during the 1970's by Barnes and Wakefield, and independently by A S Day at Ove Arup and Partners, London. It has been applied to a large number of built structures by Tensys (see Appendix A) and also by the consulting engineering companies of Ove Arup and Partners (London), Buro Happold (Bath) and IPL (Radolfzell, Germany).

Over the past two decades a number of major DR based programs have been independently developed for full engineering design and analysis purposes : by Barnes (City University), Wakefield (at Buro Happold and Tensys) and Day (at Ove Arup and Partners). These programs were developed using different programming languages and computer hardware, and each has been validated against the others.

D S Wakefield is now director of Tensys Limited, where the inTENS program suite has been successfully applied to over 700 built projects. There has been close collaboration with consulting engineers who have undertaken additional check analyses of joint project work, and who have played a major role in defining the practical engineering design requirements accommodated by the software. The output features for fabrication have been influenced by close liaison with major international membrane structure contractors, especially KOCH Membrane Structures, Architen and Landrell Fabric Engineering.

3 NUMERICAL MODELLING AND ELEMENT LIBRARY

3.1 Strut/Cable Elements

Two node line elements that may be controlled by specified tension/compression and/or elastic stiffness and slack length. For elastically controlled cable elements the current member force is monitored at all stages and reset to zero should compression be detected.

3.2 Slip Cables

A sequence of line elements through defined nodes representing a friction free cable in, for example, a pocket on the membrane surface. Under elastic control the common tension in all constituent elements is calculated from the total strain in the element sequence.

3.3 Warp Seam Stiffness

Represented by cable elements generated automatically along the geodesic seam lines. The influence of seam stiffness in reducing membrane stresses is particularly significant in the masthead ring region of conical membranes with a radial seam layout.

3.4 Membrane Elements

Triangular constant strain elements modelling orthotropic fabric behaviour. This simple element type requires a relatively dense modelling mesh but permits a simple accounting of non-linear material effects. Natural element stiffnesses (see Appendix A) are used with the convected co-ordinate system to account readily for geometric non-linearities.

The warp direction for material properties is generally aligned with the triangle side that is coincident with a geodesic seam line. The orthogonal fill, or weft, direction is then automatically set at 90 degrees to the warp.

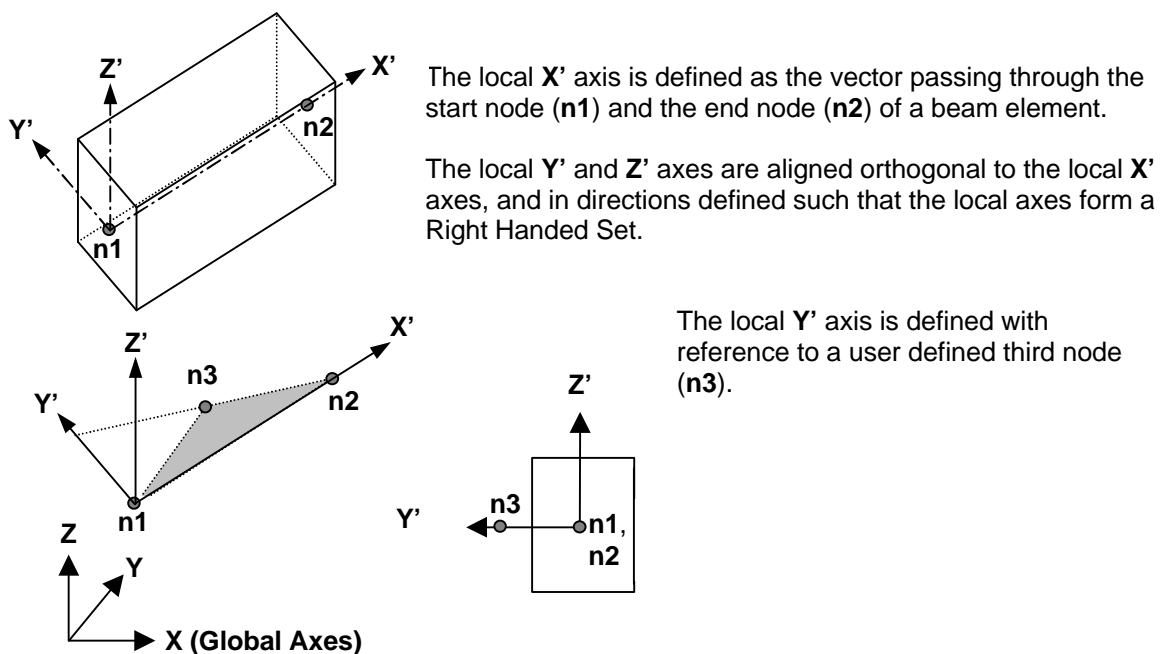
For triangles without an adjacent seam line, the warp direction is automatically set by reference to the nearest available triangle with such an alignment. During load analysis the warp and fill membrane stresses are monitored and reset to zero if any compressive stress is computed.

At the form-finding stage, stresses in the warp and fill directions are held constant within an element. Varying stresses across a surface are defined on a panel by panel basis and automatically assigned to the individual elements.

3.5 Beam Elements

Geometrically non-linear beam elements are included for enhanced support structure modelling. These are straight, two node, elements with additional rotational degrees of freedom assigned automatically. The orientation of the beam cross-section principal axes is defined by reference to a 3rd node. The influence of current axial load on the moment curvature relations is updated as an analysis proceeds by application of the established s and c stability functions. The influence of element bowing on axial load is also incorporated.

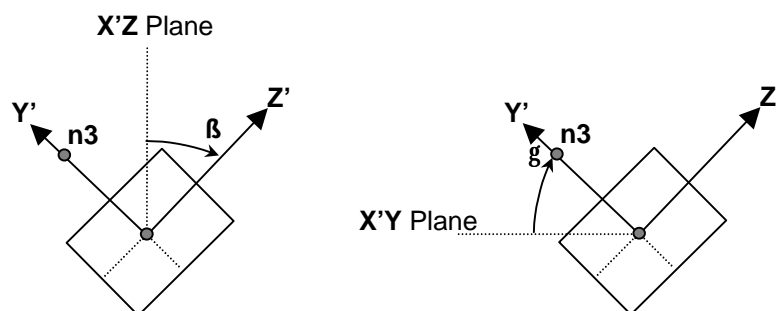
Beam Element Local Axis System



Orientation in 3D Space - Twist Angle

(i) General Case – Angle β

(ii) Special Case – Angle β
(X' & Z' Coincident)



Viewing along beam local X' axis from node $n1$ to node $n2$

4 inTENS PROGRAM MODULES

4.1 Basic Modules

TOPGEN

Topology generator for membrane surface model. Initially uses linearly approximated surface co-ordinates, but re-running TOPGEN with form found surface geometry can optimize mesh conditioning.

GENFORM

Form generation for specified membrane surface stresses. Support structure elements and restraint conditions are added at this stage. Edge cables can, for example, have specified length, dip or radius of curvature to ease adjustment specification. GENFORM is typically used iteratively, with successive refinement and adjustment of the preceding form.

ANALYS

True membrane and support structure elastic properties are introduced for load analysis of GENFORM determined form. Pre-stress levels and support structure stiffnesses and restraint conditions may be adjusted and their effects assessed without returning to the GENFORM stage.

4.2 Additional Post-Processing Modules

CUTPAT

Production of membrane cutting patterns from GENFORM output data and stretch compensation figures derived from manufacturers biannual compensation tests.

GEOMETRY

Generation of additional fabrication data, such as cable lengths and membrane plate angles. Also converts TENSYS models into .DXF format for transfer into client CAD systems.

SUMMARY

Produces reduced summary output from full load-analysis output files, which can be very large.

COLLATE

Presents selected output items (eg membrane stresses, fixed node reactions) in tabular form for all load cases with maxima and minima identified for component design.

MEMPLOT

Produces screen and hard copy graphics of analysis models at all stages.

PLATE

Produces graphic display of connection plate system geometry as DXF files.

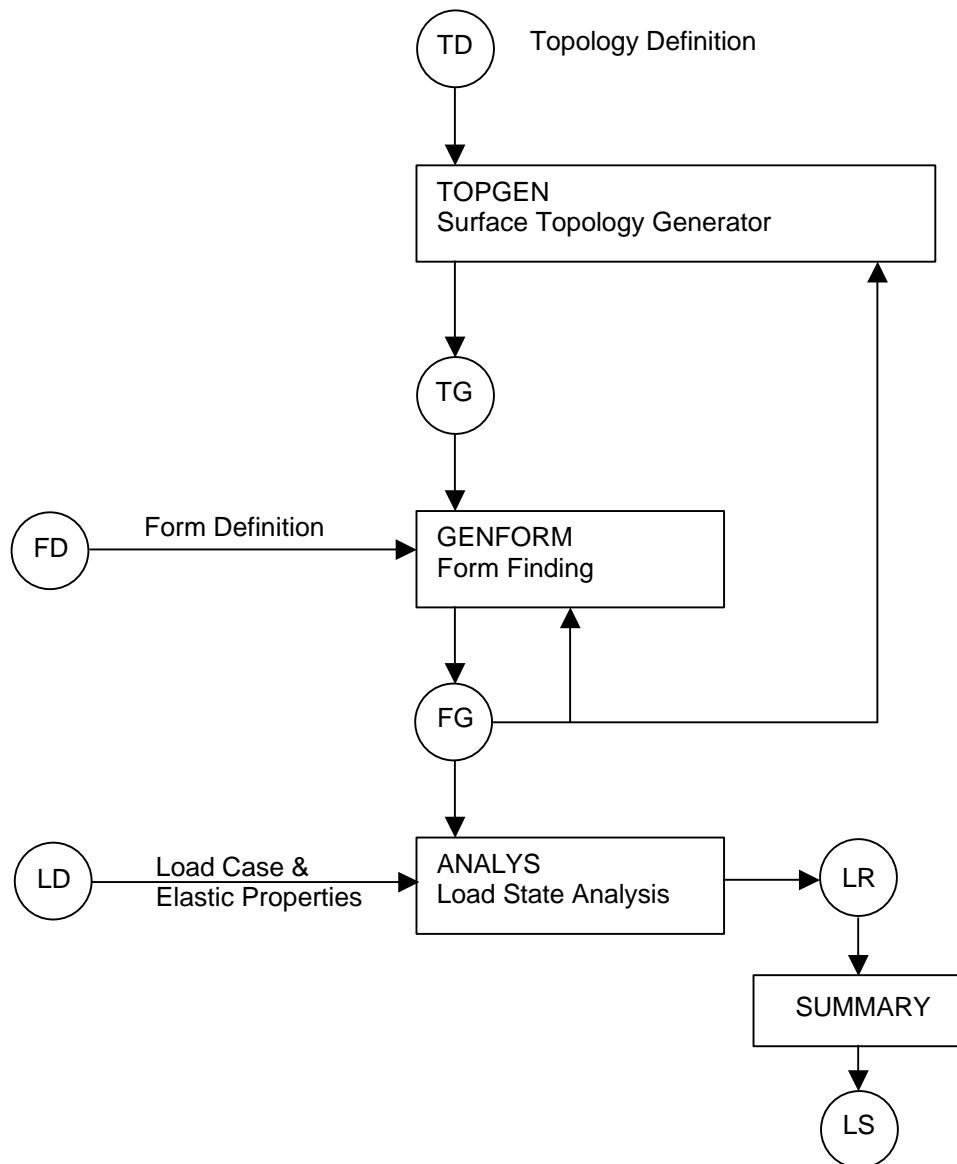
4.3 Data Handling and Program Structure

The data handling for all programs is file based. The input and output file types are:

Module	Input File(s)	Output File
TOPGEN	TD + (FG)	TG
GENFORM	FD + TG + (FG)	FG
ANALYS	LD + TG + FG	LR

Input file types TD, FD and LD are manually prepared (with details of their format given in Appendix B)

The overall data and program module flow chart is:



5 OUTPUT RESULTS AND THEIR INTERPRETATION

The full results file type LR for each load case run using ANALYS contains the following sequence of information:

5.1 Project/Problem Data

Project reference data and problem run description. Also lists name, size and creation time of input files for record purposes.

5.2 Node Co-ordinates and Residual Forces

For each node the XYZ global co-ordinates in space of the displaced structure subject to the current loading conditions are listed. The residual forces are the resultant of applied loading and member forces acting upon the node in each co-ordinate direction. At free (unconstrained) nodes these residuals should all be practically zero, giving a check on the convergence of the analysis. At restrained nodes, the residuals give the components of force imposed upon the foundation or support point by the loaded structure.

5.3 Beam End Node Rotations and Residual Moments

Displacement and residual moment output for the rotational degrees of freedom of nodes having beam elements attached. Both node rotations and residual moments (exerted by the structure upon the support point) are defined as clockwise positive about the global XYZ axes. The node rotations are effectively displacements measured from zero at the initial (unstressed) position of the beams.

5.4 Beam End Initial (Unstressed) Co-ordinates and Deflections

Again applies only to beam end nodes, if any. The unstressed Xi Yi Zi co-ordinates refer to the initially defined position of the beam ends at the TG and FD stages before any imposition of membrane pre-stress or subsequent applied loads. The listed deflections Dx, Dy and Dz are the deflections in the global co-ordinate directions of the current loaded state from this unstressed state.

5.5 Ridge/Scallop Link Tensions

Link tensions, stressed length and elastic properties, grouped together sequentially for ridge and edge scallop boundary cables. For any particular boundary the tensions in consecutive links will not usually be constant. These differences give the traction or tangential forces along the edge which must be accommodated by appropriate edge detailing such as webbing belts or cuff friction effects.

5.6 Surface Warp Cable Tensions

Tensions in cables defined along surface warp seam directions.

5.7 Support Link Tensions

Line element tensions/compressions, stressed lengths and elastic properties for group and individually defined support structure cables and struts.

5.8 Slip Cable Tensions

Cable uniform tension, elastic properties plus total and segmental stressed lengths.

5.9 Beam Element Topology

Node end connectivity for each beam element plus 3rd node definition of $x' z'$ beam local plane for bending properties and associated beam property number. The orientation of the beam local axes is also output in the traditional α -angle form as angle of twist of $x' z'$ plane from the $x' Z$ plane (or α -angle from the $x' Y$ plane when x' and Z planes are effectively coincident).

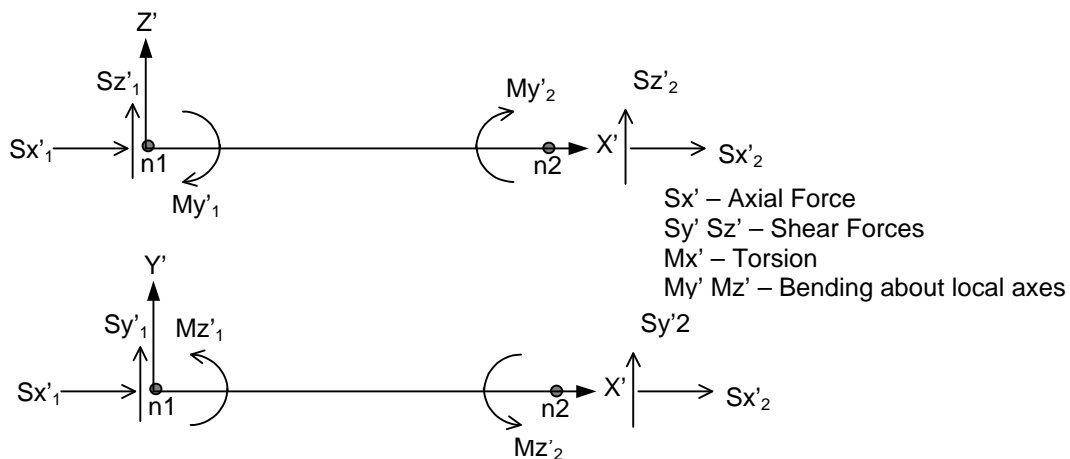
5.10 Beam Element Properties

Beam elastic properties, $EA GJ Ely' Elz'$ and self-weight. Each set is given a property number, and each beam element is thus associated with a set of properties.

5.11 Beam Element Shear Forces and Moments

Beam local axis shear and bending forces, acting on the element at the end nodes such as to cause the current state of deformation.

Sx'	Axial Force
$Sy' Sz'$	Shear Forces
Mx'	Torsion
$My' Mz'$	Bending about local (principal) axes



Beam Element Local Direct Forces and Bending Moments

5.12 Surface Warp Seam Tensions

Tension distribution along seams if stiffening effect of seam line junctions has been included.

5.13 Membrane Stress Resultants

For each region:

Initial stress for each panel. Warp and fill specified stresses from FG file, factored by any pre-stress factor applied through LD input. Material elastic properties and wind and snow surface loading definition f or the region. Current element stresses for each triangular membrane element of each panel in the region.

These are S_w and S_f , the element natural warp and fill stresses, and M_w and M_f , the element warp and fill mean of the adjacent node-averaged stresses. In the TENSYS membrane element model it is assumed that material stiffness terms are based upon biaxial tests relative to the pre-stress condition.

The stress/strain relations employed are:

S_w	Warp Stress
S_w^0	Warp Pre-stress
S_f	Fill Stress
S_f^0	Fill Pre-stress
ϵ_w	Strain from Pre-stress
ϵ_f	Strain from Pre-stress
E_{Aw}	Warp Stiffness
E_{Af}	Fill Stiffness
E_{Awf}	Cross Stiffness (Crimp Interchange Effect)

$$S_w' = S_w^0 + E_{Aw} \cdot \epsilon_w$$

$$S_w = S_w' + E_{Awf} \cdot \epsilon_f$$

$$S_f' = S_f^0 + E_{Af} \cdot f$$

$$S_f = S_f' + E_{Awf} \cdot w$$

$$\text{if } (S_w + S_f) < 0.0 \text{ } S_w = S_w' \text{ \& } S_f = S_f'$$

$$\text{if } (S_w < 0.0) \text{ } S_w = 0.0$$

$$\text{if } (S_f < 0.0) \text{ } S_f = 0.0$$

APPENDIX A COMPANY PROFILE**TENSION STRUCTURES ENGINEERING**

Tensys is an independent company providing specialist consultancy services for the engineering design of lightweight tensile structures, particularly roof constructions of stressed structural fabrics or cable networks. Munich Olympic Stadium, the Mound Stand at Lord's Cricket Ground, pavilions at Expo'92 in Seville and the new Inland Revenue Centre Amenity Building in Nottingham are well known examples of this form of construction.

The design of tensile structures requires specialist engineering skills in the development of appropriate roof forms, in the selection and application of appropriate materials and details, in the calculation of performance under environmental loading and in the generation of fabrication information. The latter includes the calculation of membrane cutting patterns and the scheduling of cable lengths and component geometry necessary for the successful realisation of these complex three dimensional roof surfaces. Numerical verification of the erection process may also be required.

Tensys was established by Dr David Wakefield to provide an independent source of these specialist design services for architects, engineers and fabricators.

The company is usually retained as a specialist consultant or sub-consultant, rather than acting as a lead consultant. In addition to the numerical analysis services, Tensys may be involved in the definition of environmental loading on tension surface structures, and also in the sizing and detailing of specific support components such as masts and connection plates. When acting for consulting engineer clients, then these clients will normally take design responsibility for these items. Foundation design, other than specification of loads and connection details will generally be undertaken by other consultants, as would the design of integrated or adjacent conventional building structures.

Tensys operates its own comprehensive computer software to facilitate the consultancy services described above. In addition colour graphics visualisation capability enables the clear presentation of projects to architect, client and planning authorities at appropriate stages of design. Dr Wakefield has been responsible for the development of tension structures CAD systems and their application to over 900 built projects. He has been involved in the development of the numerical techniques underlying this non-linear analysis software, and has a particular interest in the integration of interactive computer graphic procedures into the design process.

Tensys are able to organise wind-tunnel testing for structural loading and environmental assessment and also laboratory testing of construction materials and details. These complement the analysis service to provide a complete support package for engineers involved in a tension structure project.

The main base of the company's clients are European, with most work undertaken for British and German companies, albeit on projects that may be realised world-wide.

January 2000

Dr David Wakefield
Director

SELECTED PROJECTS

Stadia and Sports Facilities

AELTC Indoor Courts Wimbledon
 Don Valley Athletic Stadium Sheffield
 Grandstand Roof Misano Italy
 Montreal Olympic Stadium Analytical Studies
 Birkenhead Cricket Club Pavilion
 Gottlieb Daimler Stadium Stuttgart Analysis Verification
 National Hockey Stadium Milton Keynes
 Olympic Club Barcelona
 Amenity Building New Inland Revenue Centre Nottingham
 Retractable Roof Gary Weber Tennis Centre Halle
 Yacht Club Nansei Japan
 Hurlingham Covered Tennis Courts
 Taegu Stadium Korea
 Rothenbaum Stadium Hamburg

Expositions and Exhibitions

Expo'92 Seville German National Pavilion
 Oleada and Diadema Entrance Structures
 Singapore National Pavilion
 Heureka Swiss National Science Exhibition Zurich
 Ebbw Vale Garden Festival
 Press Centre World Economic Summit Munich
 Exhibition Halls Munich Exhibition Grounds
 Expo'93 Taejon S Korea German National Pavilion
 Exhibition Stands BMW Reuters BNFL Philips Peugeot
 Thai Farmers Bank Viglen Ford Audi
 GCC Conference Centre Kuwait
 Philips Pavilion Bavaria Film Park
 BMW Pavilion IAA'95 Frankfurt

Theatre and Performance Structures

International Eisteddfod Llangollen
 Retractable Theatre Roof Wiltz Luxembourg
 Rainbow Theatre Ebbw Vale Garden Festival
 Entrance Tents Pier 6 Performance Facility Baltimore
 Loro Parque Theatre Tenerife
 Eilat Creative Arts Centre Israel
 Stage Structures Michael Jackson Pink Floyd
 Paul McCartney Rolling Stones
 Shakespeare Theatre Stamford
 Buddy Holly Theatre Hamburg

Entrances Walkways and Shade Structures

Bishops Palace Excavations Witney
 Expo'92 Seville European Avenidas
 Royal Chelsea and Westminster Hospital
 Walkway Canopies Dusseldorf Airport
 Botanischer Garten Graz Austria

Mid-Kent Radiotherapy Centre
 Lampstandards Esch Luxembourg
 Train Station Euro Disney Paris
 Shade Structures Port Canaveral Florida
 Entrance Canopies Glyndebourne Opera House
 27 Broadley Terrace
 J Sainsbury Marsh Mills Plymouth
 Lords Hill Southampton
 Hilton Beach Club Dubai
 The Piazza Huddersfield
 Bradford Transport Interchange
 Castle Leisure Newport

Leisure and Recreation Facilities

Rhyl Leisure Centre
 Exchange Square London
 Bristol Zoo Restaurant Tent
 Colombo 500 Genoa
 Retractable Roof Garsington Manor Opera
 Dublin Craft Centre
 Radolfzell Gymnasium
 EPCOT Centre Future World
 Buckingham Palace Ticket Office
 Lego Park Windsor Circus Tent Music Tent Puppet Show
 Scarborough Seal Sanctuary
 Raubtierhaus Munich Zoo
 Edo Museum Tokyo
 Madagaskarhalle Zurich Zoo
 Dolphinarium and Elephant House Duisburg Zoo
 Royal Armouries Museum Leeds
 Le Coco Beach Hotel Mauritius
 Tak On Park Hong Kong
 Schildkroetenhaus Munich Zoo
 Swimming Bath Miramar Germany
 SAGA Amenity Building

Commercial and Industrial Applications

Herford Air-Supported Structures Toxic Waste Treatment
 Docklands Bridges and Landing Stage
 Health Clinic Masserberg
 Autohaus Munchen
 Swindon Covered Markets
 Stratford Bus Station
 New Inland Revenue Centre Nottingham
 Atrium Roof 71-75 Uxbridge Road
 Lloyd Studio Grid Shell
 Swindon Parade
 Water Treatment Plant Emscher
 Stairwell Roofs BHS Cambridge and Solihull
 Atrium Roof City Square Leeds
 Renault Darlaston
 Chicago Beach Tower Hotel Atrium Wall Dubai
 Yapi Kredi Bank Turkey
 Guthrie Pavilion Malaysia
 Dearne Valey College
 Dun Laoghaire Ferry Terminal
 Hospital Entrance Gaza

PUBLICATIONS

Dr David Wakefield BSc PhD CEng MIStructE MIASS
Director, Tensys Limited

'Dynamic Relaxation Analysis of Pretensioned Networks with Flexible Boundaries'
IASS World Congress, Madrid, 1979

'Dynamic Relaxation Applied to Interactive Form Finding and Analysis of Air Supported Structures'
jointly with M.R.Barnes
I Struct E Conference on Air Supported Structures, Bristol, 1984

'Computer Aided Design, Analysis and Patterning of Fabric Structures'
International Conference on Architectural Fabric Structures, Orlando, 1984

'Reconstruction of the CFAN Mission Tent'
jointly with W.I.Liddell, M.J.Cook
International Conference on Lightweight Structures, Sydney, 1986

'The Development of an Integrated CAD System for Stressed Membrane Structures'
International Conference on Lightweight Structures, Sydney, 1986

'Inverted Lattice Shell Roof, Tsim Sha Tsui Cultural Centre, Hong Kong : Design'
jointly with T.A.Ealey, P.Kelly
International Conference on Non-Linear Structures, London, 1987

'Practical Numerical Modelling of Complex Structures'
International Conference on Non-Linear Structures, London, 1987

'Form Finding, Analysis and Patterning of Surface Stressed Structures'
jointly with M.R.Barnes
I Struct E Oleg Kerensky Memorial Conference, London, 1988

'Interactive Graphic CAD for Tension Structures'
jointly with M.R.Barnes, C.F.Ong
International Conference on Civil and Structural Engineering Computing, London, 1989

'Membrane Engineering : The Role of the Specialist Engineer in the Design and Construction Process', Techtexil, Frankfurt, 1995

'Membrane Engineering : Principles and Applications'
IASS Asia-Pacific Conference on Shell and Spatial Structures, Beijing, 1996

'Membrane Engineering : Current Project Examples'
IASS International Symposium on Shell and Spatial Structures, Singapore, 1997

'Numerical Modelling in Tension Structure Design and Construction
LSA98 : Lightweight Structures in Architecture Engineering and Construction, Sydney, 1998

APPENDIX B DYNAMIC RELAXATION

The static solution of both linear and non-linear structures subject to load may be regarded as the limiting equilibrium state of damped structural vibrations excited by that load. The physical basis of Dynamic Relaxation (DR) was initially perceived as the step-by-step solution, for small time increments Δt of Newton's Second Law of Motion applied to a loaded structure subjected to an imposed viscous damping [10]. On first inspection this might seem a rather long-winded approach, but it does have significant advantages when applied to the design of tension structures.

Although originally applied to finite difference structural idealisations, the structural problem is modelled as a series of nodes at which co-ordinates are traced and masses assumed to be lumped. These nodes are interconnected by discrete finite elements representing membrane, cable, strut or beam components. At each stage of the iteration the current node residual forces are updated by considering individual elements in turn. Thus the assembly of an overall stiffness matrix is not necessary, and the specification of element prestress during form-finding or non-linear stress/strain relations during load analysis is easily achieved.

Belytschko et al [11] utilised the concept of local co-ordinates for force calculations in non-linear transient dynamic analysis, with the origin of these element co-ordinates defined by one node and the axes fixed relative to another. These local axes are thus free to translate and rotate with the structure as it undergoes deformation.

The element displacements are defined as the minimum number of geometric variables necessary to completely define the deformed configuration of that element, independent of rigid body motion which is treated automatically by the DR process. A set of associated natural element forces are determined from these basic displacements by means of the element natural stiffness relations. As well as simplifying understanding of the element, computer storage and time are saved by the minimisation of the number of degrees of freedom per element. For example, the constant strain triangular membrane element, with basic displacements defined by its three side extensions, was first proposed as an intermediate stage in a direct stiffness element derivation . It has subsequently been adopted in its natural form, with a [3x3] stiffness matrix, to the DR form-finding and analysis of prestressed membrane structures [13,14].

The iterative equations for the motion in direction x of any node i in space at time t are obtained directly from Newton (Force=MassxAcceleration) :

$$R_{ix}^t = M_i A_{ix}^t$$

which may be expressed in central difference form

$$R_{ix}^t = M_i (V_{ix}^{t+\Delta t/2} - V_{ix}^{t-\Delta t/2}) / \Delta t$$

giving the recurrence relation for nodal velocities :

$$V_{ix}^{t+\Delta t/2} = V_{ix}^{t-\Delta t/2} + \Delta t R_{ix}^t / M_i$$

where R_{ix}^t is the x direction residual force for node i at time t
 V_{ix}^t is the associated node velocity
 M_i is the node mass and Δt the time interval

The residual forces R_{ix}^t were computed for the then current node co-ordinates x_i^t . An updated set may now be calculated from the incremented node co-ordinates :

$$x_i^{t+\Delta t} = x_i^t + \Delta t V_{ix}^{t+\Delta t/2}$$

Similar recurrence relations apply to all unconstrained degrees of freedom of the structure. The stability of a time stepping dynamic analysis will depend upon the selection of a suitably small time increment, which can be shown to be a function of the local node relative stiffness [4]. When only the final static solution is sought, then fictitious nodal masses may be used that optimise convergence for a chosen time interval, which is typically taken as unity for computational efficiency. Separate masses may even be selected for each degree of freedom at a node, or square nodal mass matrices used which effectively optimise masses into the local principal stiffness directions [5]. For practical analyses, a single global mass factor is available to the user to maintain stability in cases of large initial deformation during form-finding or instability during load analysis. The detailed mass assignments are handled automatically.

The original need to compute an optimised viscous damping coefficient has been eliminated by the adoption of a kinetic damping procedure [5,15]. When an oscillating body passes through a local approximation to its static equilibrium position, then the plot of total kinetic energy against time passes through a local maximum. The total kinetic energy is traced as undamped iterations proceed, and all current node velocities are reset to zero whenever an energy peak is detected. The analysis continues, progressively eliminating the kinetic energy from various modes of vibration until the required degree of convergence is obtained. The process is automatic, requires no specification of damping factors, and can handle gross out of balance forces without the need for additional constraints. This means that gross geometrical inaccuracies and stiffness modifications may be accommodated during the formfinding process.

If one area of a particular structure should be numerically or physically unstable, then the computation will typically equilibrate the remaining structure as far as possible. Intermediate visual examination of the model during analysis, coupled with a trace on the location of current maximum residual forces facilitates diagnosis of the underlying problem.

The overall iterative process between kinetic energy peaks may be summarised :

- (a) Initialise node velocities, residuals and previous kinetic energy to zero.
- (b) Update applied loading components
- (c) Compute surface in-plane residuals for geodesic strings if membrane form-finding
- (d) Add applied loads into current residual force vector
- (e) Add resolved element force components into residual force vector
- (f) Reset residuals associated with constrained nodes to limit motion as required
- (g) Update node vectors and co-ordinates. Calculate current structure kinetic energy
- (h) If current kinetic energy > previous, then zero residual vector and return to (c)
- (i) If current kinetic energy < previous, then adjust co-ordinates to value at KE peak
- (j) Check residuals and return to (a) unless required degree of convergence obtained

The kinetic energy peak stages represent successively improving solutions to the problem, and provide a convenient point to reset element constraints and stiffnesses, for example the stability effects of current axial load on the moment-curvature relations of beam elements [16].

Coding and updating a Dynamic Relaxation program is a relatively simple and straightforward procedure. The particular advantage of the method is its clear

physical analogy and ready adaptability without compromise to the wide range of modelling problems that arise on real projects.

The following element types are available within the inTENS implementation :

Membrane Elements :

Triangular constant strain finite elements modelling orthotropic fabric behaviour. This simple element type requires a relatively dense modelling mesh but permits a straightforward introduction of non-linear material effects. Natural element stiffnesses used with the convected co-ordinate system account readily for geometric non-linearities.

The numerical form generation of membrane structures seeks an equilibrium state in which the geometry and member forces of a structure are known. Material selfweight is generally neglected from this process, and the actual material properties of the elements need not be pre-assigned. The shapes defined are a function of relative stresses, and thus even the member forces may be scaled from the final desired prestress level.

Membrane shape is controlled directly by specified warp and fill stresses in constant stress triangular finite elements. Stresses may be varied smoothly in both warp and fill directions to obtain the desired form. The range of possibilities can vary from the uniform stress distribution of a minimal surface, such as the New Inland Revenue Centre and Edo Tokyo Museum, to complex distributions in both warp and fill directions such as the temporary A-Klasse touring structure where the architect had clear visual and functional reasons for the particular shape he wished to achieve.

The warp direction of the membrane is generally assigned to one side of the triangular element, and controlled on the surface in an overall sense by the introduction of geodesic string trajectories. These are determined simultaneously within the form-finding computation and also fundamental to the cutting pattern post-processing option.

In some boundary areas, individual triangular elements may not have a side that is directly related to the warp direction defined by a geodesic string. In this case the element orientation is automatically assigned by reference to the nearest element in a sequence that has such an orientation.

Prior to load analysis, the true elastic properties of the membrane are introduced. The theoretical unstressed element side lengths of individual membrane elements are back calculated from the equilibrium stressed geometry and specified stresses. It is assumed that material stiffness terms are based upon biaxial tests relative to the prestress condition. The stress/strain relations employed are :

$$\begin{aligned}\sigma_w' &= \sigma_w^\circ + EA_w \cdot \epsilon_w \\ \sigma_w &= \sigma_w' + EA_{wf} \cdot \epsilon_f\end{aligned}$$

$$\begin{aligned}\sigma_f' &= \sigma_f^\circ + EA_f \cdot \epsilon_f \\ \sigma_f &= \sigma_f' + EA_{wf} \cdot \epsilon_w\end{aligned}$$

$$\begin{aligned}\text{if } (\sigma_w + \sigma_f < 0.0) \quad &\sigma_w = \sigma_w' \quad \& \quad \sigma_f = \sigma_f' \\ \text{if } (\sigma_w < 0.0) \quad &\sigma_w = 0.0 \\ \text{if } (\sigma_f < 0.0) \quad &\sigma_f = 0.0\end{aligned}$$

where σ_w warp stress σ_w° warp prestress
 σ_f fill stress σ_f° fill prestress
 $\epsilon_w \epsilon_f$ warp & fill strains measured from prestress state
 $EA_w \quad EA_f \quad EA_{wf}$ warp fill and cross stiffness

The membrane elements themselves provide a convenient reference for the calculation of plan and surface applied loading. Pressure loading, whether due to snow, wind suction or pressure, or internal inflation pressure may be conveniently updated according to current geometry at kinetic energy peak reset stages as the analysis proceeds.

Geodesic Strings :

Geodesic trajectories are determined by introducing elastically or tensioned controlled strings into the membrane surface. By introducing these strings with high tensions in such a way that they only control the in-plane movement of surface nodes, then the strings take up geodesic lines whose orientation is governed by their boundary end positions. Elastic control can be used to control the spacing of nodes along the string, whilst their out-of plane residual force components are suppressed so as not to influence the surface shape. Residual forces are also neglected at string ends so as not to influence boundary forces or edge cable geometries. This procedure was originally developed so as to maintain stability of a model during form-finding with non-uniform warp and fill stresses. The numerical procedure for simultaneous geodesic calculation fits into stage (c) of the calculation loop as described above :

1. calculate geodesic string residuals $\{Rg\}$ before all other element types
2. calculate current node unit surface normal vectors $\{Vn\}$ from the mean of normal vectors of adjacent triangular membrane elements weighted in inverse proportion to their area
3. thus the normal, or out-of-plane, residual component at a node is $Rn = \{Vn\}^T \{Rg\}$
4. for each geodesic string node, subtract the normal component from the global residuals : $\{R\} = \{R\} - Rn \{Vn\}$
5. set global residuals of string end nodes to zero
6. continue residual summation for other element types

Geodesic strings are only active at the form-finding stage, although their topologies are used at the cutting pattern output stage.

Strut/Cable Elements :

Two node line elements which may be controlled by specified tension/compression and/or elastic stiffness and slack length.

The form of boundary cables may be controlled by length or force. For length control, it is computationally most convenient to assign an (arbitrary) elastic property and unstressed length. For force control, one line element in the sequence making up a boundary is assigned a specific tension which is maintained throughout the iterative process.

For elastically controlled cable elements the current member force is monitored at all stages and reset to zero should compression be detected.

Slip Cables :

A sequence of line elements through defined nodes representing a friction-free cable in, for example, a pocket on the membrane surface. Under elastic control the common tension in all constituent elements is calculated from the total strain in the element sequence. The uncoupled nature of the DR process makes it particularly straightforward to implement elements of this type.

Warp Seam Stiffness :

Modelled by cable elements generated automatically along the geodesic seam lines. The influence of seam stiffness in reducing membrane stresses is particularly significant in the high point ring region of conical membranes with a radial seam layout (for example the outer membrane of Rothenbaum Stadium).

Beam Elements :

Geometrically non-linear beam elements are included for enhanced support structure modelling. These are straight, two node, elements with additional rotational degrees of freedom assigned automatically. The orientation of the beam cross-section principal axes are defined by reference to a third node. The influence of current axial load on the moment curvature relations is updated at kinetic energy peak stages as the analysis proceeds by application of the established s and c stability functions. The influence of element bowing on axial load is also incorporated.

Bending stiff beam elements may also be included in the form-generation process. In this case, however, real elastic properties should be used together with the desired rather than arbitrary levels of prestress in cables and membranes. An unstressed geometry for the beam elements must be defined as the starting point of the analyses, and this will be the reference for all subsequent force calculations in both form-finding and load analysis.

Examples of the integrated use of bending elements include the Nottingham Inland Revenue Centre, SAGA Amenity Building and the Atrium Wall for the Chicago Beach Tower Hotel development in Dubai.

Node Constraints :

Node constraints may be readily implemented into the DR iteration scheme by either resetting current velocities to zero for a fully fixed degree of freedom, or resetting node co-ordinates according to a specific constraint requirement. This is undertaken at every iteration step as outlined in section 4 above.

Translational fixity in global directions and rotational fixity about global and arbitrary axes are available. In addition nodes may be constrained to lie within arbitrary planes or on arbitrary vectors. As these planes and vectors are defined in terms of structure nodes, they can also move in space as the structure deforms if necessary. The movement of unconnected fabric over supporting arches is currently represented by the use of spherical constraint surfaces, and this principle may readily be extended to other geometric forms.

These constraint planes and surfaces may also be set to allow lift-off of the nodes in one of the local surface perpendicular directions if the current residual force components are positive in that direction. Thus for the arch example, one can model possible fabric lift-off under wind suction. This was utilised in the load analysis of the Kawakita Ohasi Toll Gate project, where the membrane was supported on a series of discrete discs parallel to the dome local surface.

At the form-finding stage some manual intervention may be necessary for on-off contact problems. For the Edo Tokyo museum a PTFE glass membrane was stressed over a series of skeletal steel dome shapes, both bearing on the curved tubular members and then spanning freely between them. The modelling of the critical transition zones where the fabric lifted off an arch was handled iteratively by the examination of local graphics and node residual components.

Cutting Patterns and Fabrication Geometry

The final fabrication geometry for all aspects of a tension structure project should be derived from a single computer model. This is the final form model that includes the detailed geometry of all system points and connection details as taken from the final fabrication drawings. The geodesic seam lines on the membrane surface will have been adjusted such that their relative spacing does not exceed the useable width of fabric. The three dimensional geometry of a particular fabric panel is then flattened out to give the pattern shape by successively unfolding the triangular elements between adjacent warp lines. In regions of high transverse curvature a better representation may be made by subdividing a physical panel into separate panels in the computer model. These are unfolded individually into a plane and subsequently sheared together at their common boundary. This is physically admissible as a solution because of the low shear stiffness of coated fabrics.

On unfolding the patterns still represent the final stressed geometry of the membrane. They must be compensated to allow for the actual stretching of the material from the unstressed as-cut state to the installed and stressed condition. The necessary warp and fill direction compensation factors are obtained from biaxial testing of the actual production material to be used. These tests will account for an initial construction stretch of the material, for the level of prestress to be utilised and for creep due to temperature and applied loading. Reduced compensation, or decompensation, may be applied at edge details to ease installation where the stiffness of local members is greater than that of the fabric.

As well as patterning the membrane, boundary and support cable element lengths, pretensions and end fittings must be scheduled, together with the final geometry and orientation of connection elements such as masthead details and steel corner plates.

References

- [1] Otto F - Tensile Structures ; *MIT Press 1971*
- [2] Wakefield DS - Membrane Engineering : Principles and Applications ; *IASS Asia Pacific Conference on Shell and Spatial Structures Beijing 1996*
- [3] Wakefield DS - Membrane Engineering : Current Project Examples ; *IASS International Symposium on Shell and Spatial Structures Singapore 1997*
- [4] Barnes MR - Form-finding and Analysis of Tension Structures by Dynamic Relaxation ; *PhD Thesis The City University London 1977*
- [5] Wakefield DS - Pretensioned Networks Supported by Compression Arches ; *PhD Thesis The City University London 1980*
- [6] Bucholdt HA & McMillan BR - Iterative Methods for the Solution of Pretensioned Cable Structures ; *IASS Symposium on Tension Structures and Space Frames Tokyo 1971*
- [7] Schek HJ - The Force Density Method for Form-finding and Computation of General Networks ; *Comp. Meth. Appl. Mech. Eng, 3 1974*
- [8] Gründig L & Bäurle J - Automated Cutting Pattern Determination and Control for Prestressed Membranes ; *Symposium on Textile Composites in Building Construction Lyon 1990*

- [9] Maurin B & Motro R - Density Methods and Minimal Forms Computation ; *IASS Colloquium on Structural Morphology Nottingham 1997*
- [10] Day AS - An Introduction to Dynamic Relaxation - *The Engineer January 1965*
- [11] Belytschko T et al - Efficient Large Scale Non-linear Transient Analysis by Finite Elements ; *IJNME 10 1976*
- [12] Argyris JH et al - A General Method for the Shape Finding of Lightweight Structures ; *Int Conference on Tension Structures London 1974*
- [13] Barnes MR - Applications of Dynamic Relaxation to the Design and Analysis of Cable, Membrane and Pneumatic Structures ; *Int Conference on Space Structures Guildford 1975*
- [14] Barnes MR - Form and Stress Engineering of Tension Structures ; *Structural Engineering Review 6 3-4 1994*
- [15] Barnes MR & Wakefield DS - Form-finding, Analysis and Patterning of Surface Stressed Structures ; *1st Oleg Kerensky Memorial Conference London 1988*
- [16] Wakefield DS - Dynamic Relaxation Analysis of Pretensioned Networks with Flexible Boundaries ; *IASS World Congress Madrid 1979*

APPENDIX C DATA FILE STRUCTURES

Module	File	Release
TOPGEN	TD	3.09
GENFORM	FD	3.22
ANALYS	LD	3.30
CUTPAT	CD CP	3.19 3.19
MLOT		4.11
GEOMETRY	SL XD	3.06 3.06
TOPMOD	TM	3.05

DATA

- File Description Line 1 (50 chars max)
- File Description Line 2 ..

- nsp nssl ncsl nasl nrl npsl ncomsl nreg

- iclock (1,nreg)

- n x y z [1,nsp]

il n1 n2 nseg nns [1,nssl]
 {if il (-)ve}
 bl(1,nseg)

DESCRIPTION

FILE DESCRIPTORS

first two lines of TD file are read as text for automatic inclusion as TG run description

SYSTEM NUMBERING

nsp no. of system points
 nssl no. of straight system lines
 ncsl no. of circular system lines (total)
 nasl no. of arbitrary system lines
 nrl no. of active ridge/scallop lines
 npsl no. of passive system lines
 ncomsl no. of composite system lines
 nreg no. of regions

MEMBRANE ORIENTATION

iclock() membrane orientation for each region
 enter ... +1 for clockwise
 -1 for anticlockwise

SYSTEM NODE CO-ORDINATES

n system point node no.
 x y z x, y, z co-ordinates

STRAIGHT SYSTEM LINES

il system line no.
 n1 n2 end system nodes
 nseg no. of segments
 nns first inter node no. (0 for automatic)
 bl() segment relative lengths

DATA

nct1 nct2

il n1 n2 n3 nseg nns [1,nct1]
 {if il (-)ve}
 bl (1,nseg)

il n1 n2 n3 n4 rad nseg nns [1,nct2]
 {if il (-)ve}
 bl (1,nseg)

il ntyp ncp nseg nns [1,nas]
 cp (1,ncp)
 {if ntyp=2,3}
 nin (1,ncp-1)
 {if ntyp≠1 and il (-)ve}
 bl (1,nseg)

DESCRIPTION**CIRCULAR ARC SYSTEM LINES**

nct1 no of circular system lines defined by 3 arc node
 nct2 no defined by circle centre & adjustable radius
 (nb nct1 + nct2 = ncsl)
 il system line no.
 n1 n2 arc end nodes
 n3 intermediate system node defining arc
 nseg no. of segments
 nns first inter node no. (0 for automatic)
 bl() segment relative lengths (equal unless il (-)ve)
 n1 n2 arc end nodes ({x} updated by radius)
 n3 intermediate node ({x} calculated by TOPGEN)
 n4 node at circle centre of arc
 rad specified arc radius (radially resets n1 n2 n3 {x})

ARBITRARY SYSTEM LINES

il system line no.
 ntyp =0 interpolate new nodes between cp(1) & cp(ncp)
 =1 system line defined by control nodes only
 =2 as =1 plus spline interp'd intermediate nodes
 =3 as =2 plus linear interp'd intermediate nodes
 ncp no. of curve definition nodes
 cp() curve definition nodes
 nseg no. of segments on final system line
 nin() no. of intermediate nodes between control nodes
 nns first inter node no. (0 for automatic)
 bl() segment relative lengths (equal unless il (-)ve)
 bl() applied locally between control nodes for
 ntyp=2 & 3

DATA	DESCRIPTION
il n1 n2 nseg nns [1,nrl]	<p>ACTIVE RIDGE/SCALLOP LINES</p> <p>il system line no. n1 n2 end system nodes nseg no. of segments nns first intermediate node no. (0 for automatic)</p>
il iactive n1 n2 nseg nns [1,npsl]	<p>PASSIVE SYSTEM LINES</p> <p>il system line no. iactive associated active ridge line n1 n2 end system nodes nseg no. of segments nns first intermediate node no. (0 for automatic)</p>
il nsysl nls nlf [1,ncomsl] ncom()	<p>COMPOSITE SYSTEM LINES</p> <p>il system line no. nsysl no. of component system lines nls start local node no. on first system line nlf finish local node no. on last system line (set nls nlf = 0 for complete system line)</p> <p>ncom() system line no.s of components (set individual no.s (-)ve to change direction from initial definition)</p>

DATA

- nsqwr niwl [1,nreg]
 iws iwf l1 n1s n1f l2 n2s n2f nsegs nsegf [1,nsqwr]

iw l1 n1 l2 n2 nseg [1,niwl]

DESCRIPTION

WARP LINE DEFINITION

nsqwr	no. of sequential warp groups in region
niwl	no. of individual warp lines in region
iws	warp line start no.
iwf	warp line finish no.
l1	end one system line
n1s	local node start no. on system line one
n1f	local node finish no. on system line one
l2	end two system line
n2s	local node start no. on system line two
n2f	local node finish no. on system line two
nsegs	no. of segments on start warp line
nsegf	no. of segments on finish warp line

iw	individual warp line no.
l1	end one system line
n1	local node on system line one
l2	end two system line
n2	local node on system line two
nseg	no. of segments

DATA	DESCRIPTION
<ul style="list-style-type: none"> nsqpr nipa [1,nreg] ips lss lsf [1,nsqpr] ip ns1 ns2 [1,nipa] <ul style="list-style-type: none"> lc n1 n2 [1,ns1] lc n1 n2 [1,ns2] {if ns1=0 or ns2=0 replace lc ... by ins} 	<p>MEMBRANE PANEL DEFINITION</p> <ul style="list-style-type: none"> nsqpr no. of sequential panel zones in region nipa no. of individual panels in region ips start panel no. lss start warp line lsf finish warp line ip individual panel no. ns1 no. of system lines on side one of panel ns2 no. of system lines on side two of panel ins no. of individual node lc system line no. (+ve if warp, -ve if other) n1 start local node on lc n2 end local node on lc (if lc is +ve then n1=0 & n2=0)
<ul style="list-style-type: none"> npgtm ir ps pf nt1 nt2 [1,npgtm] 	<p>MEMBRANE TOPOLOGY MODIFIER 1</p> <ul style="list-style-type: none"> npgtm Enforces alternating triangle element unfold no. of panel groups for topology modification ir region no. ps start local panel no. pf finish local panel no. nt1 first element to be modified (local no. along panel) nt2 last element to be modified (local element nos. along panel from end 1 enter 0 for first (nt1) or last (nt2) element enter (-)ve to count back from end 2)

DATA

- nstp
ir pl nt1 nt2 [1,nstp]

DESCRIPTION

MEMBRANE TOPOLOGY MODIFIER 2

Enforces switching of adjacent triangle elements
 nstp no. of switched triangle pairs
 ir region no.
 pl local panel no.
 nt1 first element to be modified (local no. along panel)
 nt2 adjacent element to be modified
 (local element nos. along panel from end 1
 enter (-)ve to count back from end 2)

Release Notes

3.06

Details

Region boundary definition

Deleted

Membrane topology modifier 1

Enforces alternating unfold of membranes between specified elements

Release Notes

3.07

Details

Membrane topology modifier 2

Enforces switching of adjacent triangle element pairs

Release Notes

3.08

Details

File descriptors

First two lines of file must be text which is automatically assigned as the TG run description text

DATA

DESCRIPTION

Release Notes

3.09

Details

Metric / Imperial Units

Imperial Units set by entering 'Imperial' on line 4 of PROJECT file
Metric Units by entering 'Metric' on PROJECT line 4, or if line 4 not defined

DATA

- File Description Line 1 (50 chars max)
- File Description Line 2 ..

- nfn nfx nfy nfz

ncn (1,nfn)
ncx (1,nfx)
ncy (1,nfy)
ncz (1,nfz)

- ndlg
ndlng ndlmod ival value [1,ndlg]

 {if ndlng>0}
 ndno (1,ndlng)

 {if ndlng<0}
 nsno (1,ndlng)

DESCRIPTION

FILE DESCRIPTORS

first two lines of FD file are read as text for automatic inclusion as FG run description

NODE RESTRAINTS (TRANSLATIONAL)

nfn no. of fully fixed nodes
nfx no. of fixed nodes (x direction only)
nfy no. of fixed nodes (y direction only)
nfz no. of fixed nodes (z direction only)

ncn() node no.s of fully fixed nodes
ncx() node no.s of fixed nodes (x direction only)
ncy() node no.s of fixed nodes (y direction only)
ncz() node no.s of fixed nodes (z direction only)

NODE ON-OFF DISPLACEMENT CONSTRAINTS

ndlg no. of limited displacement groups
ndlng no. of nodes in limited displacement group (set (-)ve for no. of system lines)
ndlmod constraint mode for group
ndlmod=0 ... unconstrained
 1 ... XY-PLANE Zlim=max Z ordinate
 2 Zlim=min Z ordinate
 3 ... SPHERE Rs=max radius value
 4 Rs=min radius value
ival XY-PLANE Z mode : ival=0 ... absolute
 1 ... Z relative to FG
value SPHERE ival = sphere centre node number
XY-PLANE value = Zlim or Z as above
SPHERE = sphere radius Rs
(set Rs zero for calculation from FG)
ndno() node numbers in group
nsno() system lines in group (set (-)ve to ignore ends)

DATA

DESCRIPTION

- nfnr nfxr nfyр nfzr

- ncnr (1,nfnr)
ncxr (1,nfxr)
ncyr (1,nfyр)
nczr (1,nfxr)

- npax
npx nref [1,npax]

- nfnl
nsa (1,nfnl)

- nsymsl
nsa (1,nsymsl)

- nsymwl
ir nw [1,nsymwl]

NODE RESTRAINTS (ROTATIONAL)

nfnr no. of fully restrained rotational nodes
nfxr no. of nodes fixed against x-axis rotation
nfyр no. of nodes fixed against y-axis rotation
nfzr no. of nodes fixed against z-axis rotation

ncnr() node no.s of fully restrained rotational nodes
ncxr() node no.s of x-axis restrained nodes
ncyr() node no.s of y-axis restrained nodes
nczr() node no.s of z-axis restrained nodes

npax no. of arbitrary pin axis rotational restraints
npx pin restrained node
nref second node defining pin axis npx->nref

SYSTEM LINE RESTRAINTS

nfnl no. of fully fixed system lines
nsa() associated system line no.s

SYSTEM LINES ON X- OR Y- AXIS SYMMETRY LINES

nsymsl no. of system lines on axis line of symmetry
nsa() associated system line no.s

WARP LINES ON X- OR Y- AXIS SYMMETRY LINES

nsymwl no. of warp lines on axis line of symmetry
ir region no.
nw local warp line no.

DATA

DESCRIPTION

- nacp nacsl nacwl nacn

- kp it itp n1 n2 (n3) [1,nacp]

- kp imode ns [1,nacsl]

- kp imode ir nw [1,nacwl]

- kp imode ncn [1,nacn]

ARBITRARY CONSTRAINT PLANES & VECTORS

nacp no. of constraint planes/vectors
nacsl no. of constrained system lines
nacwl no. of constrained warp lines
nacn no. of constrained individual nodes
kp constraint plane/vector number
it enter it =1 ... line/vector constraint
2 ... plane constraint (2 node definition)
3 ... plane constraint (3 node definition)
(for it=2 n1 on plane and n1->n2 normal vector)
itp itp = 1 ... plane perp'r to membrane surface
0 ... otherwise
n1 n2 vector/plane definition node numbers n1 -> n2
n3 additional node for plane definition (only if it=3)
kp constraint plane/vector number
ns constrained system line
ir nw region and local number of constrained warp line
ncn node number of constrained node
imode constraint type (planes)
imode = -1 ... node on or free below plane
0 ... node held in plane
1 ... node on or free above plane
constraint type (vectors)
imode = -1 ... node on or below n1
0 ... node held on vector
1 ... node on or above n1

DATA

- ncup
n1 n2 [1,ncup]

il lg ear tr swl pval pmode smode [1,nrl]
 {if smode=1}
 bl1 bl2 bl3
 {if smode=2}
 bl (1,nseg)

DESCRIPTION

COUPLED NODES

ncup no. of coupled node pairs
 n1 master node of pair
 n2 slave node
 slave node residuals re-assigned to master
 slave node coordinates set identical to master

ACTIVE RIDGE PROPERTIES

il system line no.
 lg element group reference number
 ear EA value (MN)[kip]
 tr specified tension (kN)[kip]
 swl selfweight/unit length (kN/m)[plf]
 pval target profile value
 pmode profile mode, enter ... 0 for radius
 1 for dip 2 for length
 3 for span/dip ratio 4 for geodesic ridge
 (if 4 then set val = fictitious string tension.
 ear ignored but passed to ANALYS. tr as normal
 in addition to geodesic tension. bl as normal
 - elastically controlled geodesic)
 smode spacing mode, enter ... 0 for equal
 1 for quadratic 2 for specified
 nrl (no. of active ridge/scallop lines)
 bl1 bl2 bl3 segment relative lengths at end1 mid & end2
 bl() segment individual relative lengths
 nseg (no. of segments)

DATA

il [1,npsl]
bl (1,nreg)

- nstw
ir nw lg tws swl [1,nstw]

- ixn nslg nplg nxlg nsubl nslpg ixl nsc

nn x y z [1,ixn]

DESCRIPTION

PASSIVE SYSTEM LINES

il system line no.
npsl (no. of passive system lines)
bl () segment relative lengths

CABLE REINFORCED WARP LINES

nstw no. of cable reinforced warp lines
ir region no.
nw local warp line no.
lg element group reference number
tws specified tension (kN)[kip]
swl selfweight/unit length (kN/m)[plf]

ADDITIONAL DATA PARAMETERS

ixn no. of extra specified nodes
nslg no. of system line or sequential link traverses
nplg no. of groups of links to a point
nxlg no. of groups of node definition links with common properties
nsubl no. of subdivided links
nslpg no. of groups of links between system line pairs
ixl no. of individual extra links
nsc no. of slip cables

EXTRA NODES

nn additional node no.
x y z x, y, z co-ordinates

DATA

il lg eas ts swl blq [1, nslg]
 {if il=0}
 nlg
 nsa (1, nlg+1)

il lg ip eas ts swl bls [1, nplg]
 {if il=0}
 nlg
 nsa (1, nlg)

nlg lg eas ts swl bls [1, nxlg]
 n1 n2 [1, nlg]

DESCRIPTION

SYSTEM LINE / SEQUENTIAL LINK GROUPS

il system line no. (0 for node sequence)
 lg element group reference number
 eas EA value (MN)[kip]
 ts specified tension (kN)[kip]
 swl selfweight/unit length (kN/m)[plf]
 blq sequence total elastic length
 (set zero for calculation from node coordinates)
 nlg no. of links
 nsa() sequential nodes

GROUPS OF LINKS CONNECTED TO A COMMON POINT

il system line no. (0 for node sequence)
 (last node ignored for closed loop system line)
 ip common node no.
 lg eas ts swl as above
 bls elastic length of individual link
 (set zero for calculation from node co-ordinates)
 nlg no. of links
 nsa() sequence of remote end node nos.

GROUPS OF LINKS WITH COMMON PROPERTIES

nlg no. of links in group
 lg eas ts swl as above
 bls as above
 n1 n2 connecting nodes

DATA

n1 n2 lg nseg eas ts swl bls [1,nsubl]

nlg lg eas ts swl bls [1,nslpg]
l1 n1s n1f inc1 l2 n2s n2f inc2

n1 n2 lg eas ts swl bls [1,ixl]

DESCRIPTION**SUBDIVIDED LINKS**

n1 n2 connecting nodes
nseg no. of segments
lg eas ts swl as above
bls as above
(intermediate nodes assigned automatically)

GROUPS OF LINKS BETWEEN SYSTEM LINE PAIRS

nlg no. of links in group
lg element group reference number
eas ts swl as above
bls elastic length of individual link
(set zero for calculation from node co-ordinates)
l1 system line one
n1s local start node along system line one
n1f local finish node along system line one
inc1 increments along system line one
l2 system line two
n2s local start node along system line two
n2f local finish node along system line two
inc2 increments along system line two

INDIVIDUAL EXTRA LINKS

n1 n2 connecting nodes
lg element group reference number
eas ts swl as above
bls as above

DATA

```

il lg eas ts swl bls mode [1,nsc]
  {if il = 0}
    nseg
    nscnod (1,nseg+1)
  {if il = -1}
    ir nw

```

DESCRIPTION

SLIP CABLE DEFINITION

```

il > 0      system line number
il = 0      direct node definition
il = -1     warp line definition
lg eas ts swl  as above
bls         profile length/value
mode        profile mode, enter ...
              0 for length calculated through co-ordinates
              1 for specified length
              2 for specified dip
              3 for specified span/dip ratio
              4 for overlength slip cables (can go slack in
                Genform; set bls = slack length
                ts = zero      )

nseg        no. of cable segments
nscnod( )   node no. sequence
ir          region no.
nw          local warp number

```

DATA

- nibl nsgbl nsubb nslpbg nbprop nmbrel nbmp

n1 n2 n3 lg nbp irx iry irz [1,nibl]

isl lg ns1 nf1 n3 nbp irx iry irz [1,nsgbl]
irx1 iry1 irz1 irx2 iry2 irz2

n1 n2 n3 lg nseg nbp irx iry irz [1,nsubb]
irx1 iry1 irz1 irx2 iry2 irz2

nbg lg n3 nbp irx iry irz [1,nslpbg]
l1 n1s n1f inc1 l2 n2s n2f inc2

DESCRIPTION

BEAM DEFINITION

nibl no. of individual beams
nsgbl no. of beam sequences on a system line
nsubb no. of subdivided beams between node pairs
nslpbg no. of beam sequences between system line pairs
nbprop no. of beam properties
nmbrel no. of modified beam internal releases
nbmp no. of beams with modified property nos.

n1 n2 beam end nodes
n3 node defining beam local x` y` plane
lg element group reference number
nbp beam property number
irx x' local moment release : 0 =no releases
iry y' 1 =end 1 release
irz z' 2 =end 2 release
3 =ends 1 & 2 released

isl system line
ns1 nf1 local node nos. , start and finish along system line
irx iry irz local releases as above – sequence internal beams
irx1 iry1 iry2 local releases – sequence first beam
irx2 iry2 irz2 local releases – sequence last beam

n1 n2 end nodes for subdivided beam sequence
nseg no of beams between end nodes
(intermediate nodes assigned automatically)

nbg no. of beams in group defined between sl pairs
l1 system line 1
n1s local start node along system line 1
n1f local finish node along system line 1
inc1 increments along system line 1
l2 n2s n2f inc2 as above for system line 2

DATA

DESCRIPTION

i E G A ly lz J swt [1,nbprop]

i beam property no.
 E G elastic moduli (MN/cm2)[10e6psi]
 A cross sectional area (cm2)[in2]
 ly lz J section properties (cm4)[in4]
 swt beam selfweight/unit length (kN/m)[plf]

nb irx iry irz [1,nmbrel]

nb beam no. for modified internal releases
 irx x' local moment release : 0 =no releases
 iry y' 1 =end 1 release
 irz z' 2 =end 2 release
 3 =ends 1 & 2 release

nb nbp [1,nbmp]

nb beam no. for modified property no.
 nbp new property no.

DATA

- nslbat nbatprop

- isl lg ns1 nf1 nbp ir1 ir2 blq smode [1,nslbat]
 - {if ir1=1}
 - vx1 vy1 vz1
 - {if ir2=1}
 - vx2 vy2 vz2
 - {if smode=1}
 - bl1 bl3
 - {if smode=2}
 - bl1 bl2 bl3
 - {if smode=3}
 - bl(1,nseg)

- j E A I swt [1,nbatprop]

DESCRIPTION

BATTEN DEFINITION

- nslbat no. of system line defined battens
- nbatprop no. of batten properties

- isl system line
- lg element group reference no.
- ns1 local node no., start
- nf1 local node no., finish
- nbp batten property no.
- ir1 end 1 orientation restraint 0=no 1=yes
- ir2 end 2
- {vx1 vy1 vz1} end 1 orientation vector (in direction end1->end2)
- {vx2 vy2 vz2} end 2
- blq batten total unstressed length
(set zero for calculation from node coordinates)
- smode segment relative spacing mode, enter ...
 - 0 for equal spacing 1 for linear spacing
 - 2 for quadratic 3 for specified
 - 4 for calculation from node coordinates
- nseg (no. of segments)

- j batten property no.
- E elastic modulus (MN/cm2)[10e6psi]
- A cross sectional area (cm2)[in2]
- I section properties (cm4)[in4]
- swt beam selfweight/unit length (kN/m)[plf]

DATA

- irws igws

 {if irws>0}
 tcon lg (1,nreg)

 {if igws>0}
 tcfac (1,nreg)

- nspstr

 nspstn tspst ispstmod ispstres [1,nspstr]

 nspsnod (1,nspstn)

 {if ispstmod>0}
 bl (1,nspstn-1)

- srx sry [1,nreg]

- mlmode

DESCRIPTION

WARP STRESSING / GEODESIC STRING CONTROL

irws irws=1 if regional warp stressing required
 igws igws=1 if geodesic string control required
 (constant tension coefficient)
 igws=2 if geodesic string control required
 (constant tension)
 tcon() warp line specified tension in region
 (automatically halved on symmetry line)
 lg element group reference number
 tcfac() geodesic string tension coefficient factor for
 region (or string tension if igws = 2)

★ **SPACER STRINGS**

nspstr no. of spacer strings
 nspstn no. of nodes on spacer string
 tspst spacer string control tension
 ispstmod spacing mode 0 = equal 1 = specified
 ispstres residual mode 0 = geodesic 1 = tangential
 nspsnod() node sequence along spacer string
 bl() segment relative spacing

DEFAULT MEMBRANE STRESS

srx membrane stress in warp direction (kN/m)[pli]
 sry membrane stress in fill direction (kN/m)[pli]

MEMBRANE REGION BOUNDARY LOADING OPTION

mlmode =0 surface swt and loading distributed via all nodes
 1 no loading at cable,beam & fully fixed nodes
 2 as option 1 with loads redistributed in element
 3 area redistribution of load across region
 (normally set mlmode = 3)

DATA	DESCRIPTION
<ul style="list-style-type: none"> swm(1,nreg) 	<p>MEMBRANE SELFWEIGHT</p> <p>swm() region membrane selfwt/unit area (kN/m2)[oz/yd2] (normally zero for form finding)</p>
<ul style="list-style-type: none"> npgvws ir ps pf nt1 nt2 sw1 sw2 sw3 fw1 fw2 fw3 [1,npgvws] 	<p>MEMBRANE PANEL ZONES WITH VARYING WARP STRESS</p> <p>npgvws no. of panel sequences with varying warp stress ir region no. ps start panel no. pf finish panel no. nt1 zone end one local triangle no. along panel nt2 zone end two local triangle no. along panel (enter 0 for panel first and/or last triangles) sw1 zone end one warp stress at start (kN/m)[pli] sw2 mid zone warp stress at start sw3 zone end two warp stress at start fw1 zone end one warp stress at finish fw2 mid zone warp stress at finish fw3 zone end two warp stress at finish</p> <p>stresses vary quadratically at each triangle</p>

DATA

- ncswpg
 ir p1 nt11 nt12 p2 nt21 nt22 ncpt [1,ncswpg]
 - swi swil [1,ncpt]
 - sfi sfil [1,ncpt]

- npgvfs
 ir ps pf nt1 nt2 sf1 sf2 sf3 ff1 ff2 ff3 [1,npgvfs]

DESCRIPTION**MEMBRANE WARP STRESS : CUBIC SPLINE VARIATIONS**

ncswpg no. of panel sequences with splined warp stress
 ir region no.
 p1 start panel no.
 nt11 zone end one local triangle no. along start panel p1
 nt12 zone end two local triangle no. along start panel p1
 (enter 0 for panel first and/or last triangles)
 p2 finish panel no
 nt21 zone end one local triangle no. along finish panel p2
 nt22 zone end two local triangle no. along finish panel p2
 ncpt no of control points for start and finish panel splines
 swi start panel spline data : stress swi at control point i
 swil relative length of control point i along panel
 fwi finish panel spline data : stress fwi at control point i
 fwil relative length of control point i along panel

stresses vary by cubic spline at each triangle
 control data interpolated linearly for intermediate panels

MEMBRANE PANEL ZONES WITH VARYING FILL STRESS

npgvfs no. of panel sequences with varying fill stress
 ir region no.
 ps start panel no.
 pf finish panel no.
 nt1 zone end one local triangle no. along panel
 nt2 zone end two local triangle no. along panel
 (enter 0 for panel first and/or last triangles)
 sf1 zone end one fill stress at start (kN/m)[pli]
 sf2 mid zone panel fill stress at start
 sf3 zone end two fill stress at start
 ff1 zone end one fill stress at finish
 ff2 mid zone fill stress at finish
 ff3 zone end two fill stress at finish

quadratically varying stresses applied in blocks
 of two triangles starting from end one

DATA

DESCRIPTION

- infl
pinfl (1,nreg)

- nload
{if nload>0}
nn px py pz [1,nload]

INFLATION PRESSURE

infl infl = 1 if regional inflation pressure required
 0 otherwise
pinfl() regional inflation pressure (kN/m2)[psf]

NODE POINT LOADS

nload no. of loaded nodes
nn node no.
px py pz load (kN)[kip] in global x y z directions

Release Notes

3.18

Node Constraint Added

Details

Constrain nodes onto specified system or warp line

Release Notes

3.19

File descriptors

Beam Definitions

Details

First two lines of file must be text which is automatically assigned as the FG run description text

Additional release options added to system line defined beam sequence
Subdivided beam sequence between node pair added

Release Notes

3.20

Metric / Imperial Units

Details

Imperial Units set by entering 'Imperial' on line 4 of PROJECT file
Metric Units by entering 'Metric' on PROJECT line 4, or no line 4 defined

DATA**DESCRIPTION****Release Notes****3.21**

Varying Warp Stresses

Details

Option added for cubic spline defined varying warp stresses

Release Notes**3.22**

Spacer Strings

Details

String sequence to adjust surface node spacing. Equal or relative spacing may be imposed, with tension control regulating strength of control. Strings behave like warp lines, with no residual impact on end nodes and out of plane components deleted. Tangent residual option further restricts influence of residuals to tangent direction defined by 2 adjacent nodes. String tensions will be redistributed to membranes at ANALYS stage.

DATA

- File Description Line 1 (50 chars max)
- File Description Line 2 ..

- nnrel nslrel ncfix

- nrn (1,nnrel)

- nrsl (1,nslrel)

- nc ix iy iz [1,ncfix]

DESCRIPTION

FILE DESCRIPTORS

first two lines of LD file are read as text for automatic inclusion as LR run description

NODE RESTRAINT MODIFICATIONS (relative to FG state)

nnrel	no. of individual nodes to be released
nslrel	no. of system lines to be released
ncfix	no. of nodes with revised fixity conditions
nrn()	node numbers to be released
nrsl()	system line numbers to be released set negative for internal nodes only released
nc	node number for revised restraint condition
ix iy iz	0=free 1=fixed in X Y Z directions

DATA

- indlg
 - {if indlg=1}
 - ndlg
 - ndlng ndlmod ival value [1,ndlg]
 - {if ndlng>0}
 - ndno (1,ndlng)
 - {if ndlng<0}
 - nsno (1,ndlng)

DESCRIPTION

NODE ON-OFF DISPLACEMENT CONSTRAINTS

indlg	indlg=0 ... utilise displacement limits from FG 1 ... read new data from below 2 ... cancel displacement limits from FG
ndlg	no. of limited displacement groups
ndlng	no. of nodes in limited displacement group (set (-)ve for no. of system lines)
ndlmod	constraint mode for group ndlmod=0 ... unconstrained 1 ... XY-PLANE Zlim=max Z ordinate 2 Zlim=min Z ordinate 3 ... SPHERE Rs=max radius value 4 Rs=min radius value
ival	XY-PLANE Z mode : ival=0 ... absolute 1 ... Z relative to FG SPHERE ival = sphere centre node number
value	XY-PLANE value = Zlim or Z as above SPHERE = sphere radius Rs (set Rs zero for calculation from FG)
ndno()	node numbers in group
nsno()	system lines in group (set (-)ve to ignore ends)

DATA	DESCRIPTION
<ul style="list-style-type: none"> nracp nracsl nracwl nracn 	<p>PLANE / VECTOR CONSTRAINT MODIFICATIONS</p> <p>nracp no. of constraint planes/vectors to be deleted nracsl no. of system lines to be unconstrained nracwl no. of warp lines to be unconstrained nracn no. of individual nodes to be unconstrained kpa() constraint plane/vector numbers for deletion</p> <p>nsa() system line numbers for unconstraint ir nw region and local warp line for unconstraint ncn() individual node numbers for unconstraint</p>
<p>kpa (1,nracp)</p> <p>nsa (1,nracsl) ir nw [1,nracwl] ncn (1,nracn)</p>	
<ul style="list-style-type: none"> nrslm nrinm 	<p>MIRROR REFLECTION MODIFICATIONS</p> <p>nrslm no. of mirrored system lines to be unconstrained nrinm no. of mirrored nodes to be unconstrained</p> <p>nursl() system line numbers for unconstraint nurin() individual node numbers for unconstraint</p>
<p>nursl (1,nrslm) nurin (1,nrinm)</p> <p>{if nrsl≠0} iflag</p> <p>{if iflag=1} eas swl (1,nrsl)</p> <p>{if iflag=2} eas (1,nrsl)</p>	<p>RIDGE / SCALLOP LINES</p> <p>iflag iflag=0 if eas and swl to be read from FG file iflag=1 if eas and swl to be defined in LD file iflag=2 if eas to be defined in LD file and swl to be read from FG file</p> <p>nrsl (no. of ridge/scallop lines) eas() EA value (MN)[kip] swl() selfweight/unit length (kN/m)[plf]</p>
<p>{if nstw>0} eas swl (1,nstw)</p>	<p>CABLE REINFORCED WARP LINES</p> <p>nstw (no. of cable reinforced warp lines as defined in FD file) eas() swl() as above</p>

DATA

{if nslg nplg nxlg nsubl nslpg >0}
iflag

{if iflag=1}
eas swl (1,nslg)
eas swl (1,nplg) etc for active groups

{if iflag=2}
eas (1,nslg)
eas (1,nplg) etc

{if ixl>0}
iflag

{if iflag=1}
eas swl (1,ixl)

{if iflag=2}
eas (1,ixl)

- nslmp
j eas(j) swl(j) [1,nslmp]

DESCRIPTION**ADDITIONAL CABLE GROUPS**

nslg etc (additional cable groups as defined in FD file)
iflag iflag=0 if group eas and swl to be read from
FG file
iflag=1 if group eas and swl to be defined in
LD file

iflag=2 if group eas to be defined in LD file and
swl to be read from FG file
eas() EA value (MN)[kip]
swl() selfweight (kN/m)[plf]

INDIVIDUAL EXTRA LINKS

ixl (no. of extra links as defined in FD file)
iflag iflag=0 if link eas and swl to be read from
FG file
iflag=1 if link eas and swl to be defined in
LD file
iflag=2 if link eas to be defined in LD file and
swl to be read from FG file
eas() swl() as above

SPECIFIED LINK PROPERTY MODIFICATIONS

nslmp no. of specified links
j link no.
eas(j) swl(j) as above

DATA

{if irws>0}
eas (1,nreg)

{if nsc>0}
iflag

{if iflag=1}
eas swl (1,nsc)

{if iflag=2}
eas (1,nsc)

{if nbprop>0}
nvbp

{if nvbp>0}
i E G A ly lz J swt [1,nvbp]

DESCRIPTION**WARP SEAM LINES**

irws (regional warp stressing as defined in FD file)
nreg (no. of regions)
eas() as above

SLIP CABLES

nsc (no. of slip cables as defined in FD file)
iflag iflag=0 if cable eas and swl to be read from
FG file
iflag=1 if cable eas and swl to be defined in
LD file
iflag=2 if cable eas to be defined in LD file and
and swl to be read from FG file
eas() EA value (MN)[kip]
swl() selfweight (kN/m)[plf]

BEAM PROPERTIES

nbprop (no. of beam properties as defined in FD file)
nvbp nvbp=0 take FD beam properties from FG file
nvbp>0 no of beam property revisions
(equilibrium will not be retained)
i beam property no. for revision
E G elastic moduli (MN/cm²)[10e6psi]
A cross sectional area (cm²)[in²]
ly lz J section properties (cm⁴)[in⁴]
swt beam selfweight/unit length (kN/m)[plf]
(setting E G zero effectively deletes beam)

DATA	DESCRIPTION
<ul style="list-style-type: none"> • imode {if imode=2,3or4} irflag (1,nreg) • EAw EAf EAwf G Swm [1,nreg] • npfk nr np feaw feaf feawf fg [1,npfk] • ipmode {if ipmode=1} PD_filename {if ipmode=2} pfill 	<p>MEMBRANE ANALYSIS MODE</p> pmode = 0 normal membrane mode = 1 all membranes & surface loading ignored = 2 regional deletion irflag(ir) = 0 deleted = 1 active = 3 homogeneous material - compression allowed irflag(ir) = 0 (normal) = 1 (homogeneous) = 4 homogeneous material - no compression irflag(ir) = 0 (normal) = 1 (homogeneous)
	<p>MEMBRANE ELASTIC PROPERTIES</p> nreg (no. of regions) EAw E value in warp direction (MN/m)[klf ... kip/ft] EAf E value in fill direction (MN/m)[klf] EAwf cross stiffness (MN/m)[klf] G shear stiffness (MN/m)[klf] Swm membrane selfweight (kN/m ²)[oz/yd ²] (no entry if imode = 1)
	<p>MEMBRANE PANEL STIFFNESS FACTORS</p> npfk no. of panels with factored stiffnesses (applied to regional properties above) nr region no. np local panel no. feaw feaf factors applied to Eaw and Eaf feawf fg factors applied to Eawf and G
	<p>PRESTRESS DISTRIBUTION</p> ipmode prestress distribution mode 0 = FD/FG arrangement 1 = new distribution via PD file 2 = force specified fill stress throughout 3 = output membrane unstressed geom'y to MS 4 = input unstressed geometry from MS file

DATA	DESCRIPTION
<ul style="list-style-type: none"> • sfac 	<p>PRESTRESS FACTOR</p> <p>sfac prestress factor (applied to prestress levels set in FD/FG files)</p>
<ul style="list-style-type: none"> • itemp <p>{if itemp=1} alphas dtheta</p>	<p>TEMPERATURE DIFFERENTIAL</p> <p>itemp itemp=1 if temperature differential to be applied to links slip cables and beams itemp=0 if no temperature differential</p> <p>alphat coefficient of linear thermal expansion dtheta temperature increase (°C)[°F]</p>

DATA

- ipld
 {if ipld≠0}
 npblok
 bzmin bzmax qzmin qzmax [1,npblok]

- {if ipld≠0}
 nspfg
 iwmode
 nr i1 i2 fac1 fac2 fac3 [1,nspfg]

- {if ipld≠0}
 ncspfg
 nr i1 i2 nt1 nt2 ncpt ibt1 [1,ncspfg]
 cpp cplen (1,ncpt)

- [**nb** Negative Pressure Factors indicate Suction
 on membrane element top surface

- loading is additive when multiple coefficient
 definitions made for a given panel]

DESCRIPTION

EXTERNAL PRESSURE LOADING

- ipld ipld = 1 pressure applied via membrane elements
 ipld = 0 no external pressure applied

- npblok no. of reference pressure definition blocks
- bzmin block lower z level zmin
- bzmax block upper z level zmax
- qzmin ref pressure q (kN/m2)[psf] at block lower z level
- qzmax upper z level
 (q linearly interpolated between qzmin & qzmax
 for z≥zmin and z<zmax)

- nspfg no. of simple panel factor definition groups
- iwmode factor interpolation mode :
 0 = linear interp'n end1->mid & mid->end2
 1 = quadratic interp'n through end1,mid,end2

- nr region no.
- i1 start panel no.
- i2 finish panel no.
- fac1 pressure factor panel end one
- fac2 pressure factor mid panel
- fac3 pressure factor panel end two

- ncspfg no. of cubic spline factor definition groups
- nr region no..
- i1 start panel no.
- i2 finish panel no.
- nt1 zone end one local triangle no. along panel
- nt2 zone end two local triangle no. along panel
 (enter 0 for panel first and/or last triangles)

- ncpt no. of reference coefficients for spline definition
- ibt1 zone start block (0=partial 1=two triangles)
- cpp() reference pressure factors along zone
- cplen() reference point relative lengths along zone

- spline varying factors applied in blocks of two
 triangles starting from end one according to ibt1

DATA

- isnow

{if isnow=1}

snow

nsfpg

nr i1 i2 nt1 nt2 fac1 fac2 [1, nsfpg]

{if isnow=2}

$\alpha 1$ $\alpha 2$ ks0 ks1 ks2 ks3

rsnow (1,nreg)

DESCRIPTION

SNOW LOADING

- isnow isnow=0 if no snow loading required

isnow=1 if panel zone defined snow loading

isnow=2 if slope defined snow loading

snow basic snow load (kN/m2)[psf] on plan area

nsfpg no. of snow factor panel groups

nr region no.

i1 start panel no.

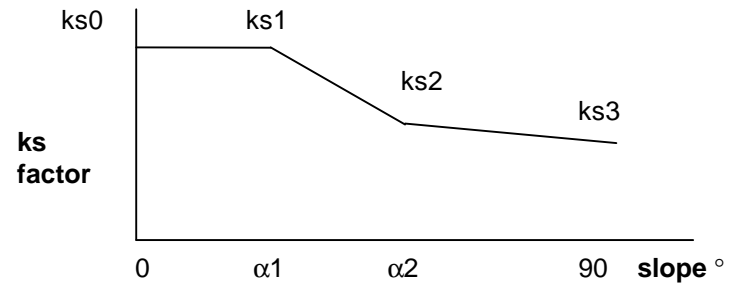
i2 finish panel no.

nt1 zone end 1 local triangle no. along panel

nt2 zone end 2 local triangle no. along panel
(enter 0 for panel first and/or last triangles)

fac1 snow pressure factor zone end 1

fac2 snow pressure factor zone end 2



- rsnow() basic snow load / region (kN/m2)[psf] on plan area
(applied snow() = ks x rsnow())

[**nb** for (+)ve rsnow() and fac1,fac2
snow load applied in Z (-)ve direction]

DATA

- nlex

{if nlex>0}
nlx n1 n2 mode dl [1,nlex]

- nbshift

{if nbshift>0}
nbs nbt [1,nbshift]

- nlten

{if nlten>0}
nlx n1 n2 ten [1,nlten]

- nscten

{if nscten>0}
nscx ns nf ten [1,nlten]

DESCRIPTION**SPECIFIED LINK ELEMENT EXTENSIONS/LENGTHS**

nlex	no. of links to be extended
nlx	link no.
n1 n2	node nos.
mode	mode=0 set dl = link extension relative to FG (dl negative for link shortening) mode=1 set dl = new link length at FG stress

BOUNDARY INITIAL COORDINATE SHIFT

nbshift	no. of boundaries to be shifted
nbs	boundary no. to be shifted
nbt	target boundary no. for shift nodes of boundary nbs shifted to node coordinates of boundary nbt (number of nodes in each boundary pair must be equal node fixity must be assigned as normal)

SPECIFIED LINK ELEMENT TENSIONS

nlten	no. of specified tension links
nlx	link no.
n1 n2	node nos.
ten	specified tension (kN)[kip] (set negative for compression)

SPECIFIED SLIP CABLE TENSIONS

nscten	no. of specified tension slip cables
nscx	slip cable no.
ns nf	start and finish node nos.
ten	specified tension (kN)[kip] (set negative for compression)

DATA

- ncell mode

- {if mode=2}
 maxcycle pincmax ptol ivmode
 pcellest pcellmin pcellmax [1,ncell]
 { if ivmode=1 }
 vcell (1,ncell)

- ncreg pcell zcell ulift [1,ncell]
 nr norient [1,ncreg]

DESCRIPTION

- * **GAS CELL DEFINITION & CONTROL**
- ncell no. of closed gas cells
- mode mode = 0 cells inactive
 mode = 1 constant internal pressure
 mode = 2 constant weight (P.V constant)
- maxcycle max no. of analysis cycles
- pincmax max cell pressure increment between cycles
- ptol internal pressure convergence tolerance
- ivmode cell ref volume mode
 ivmode = 0 calculate from LR start geometry
 ivmode = 1 input cell ref volumes vcell()
- pcellest estimated final cell pressure
- pcellmin estimated cell minimum pressure
- pcellmax estimated cell maximum pressure
- ncreg no of regions defining cell
- pcell cell ref internal pressure above atm (kN/m2)[psf]
- zcell reference elevation for zero differential pressure
- ulift unit lift (kN/m3)[lbf/ft3] for differential gas pressure
- nr region no.
- norient region orientation relative to cell :
 norient = 1 membrane top on cell exterior
 norient =-1 interior

- cell reference internal pressure associated with equilibrium form from FG (defined inflation) or from initial LR form (mode=1)

- for mode=2 cell pressure adjusted automatically at end of each converged cycle to satisfy constant weight requirement until adjustment less than ptol

- pcellest used for first cycle of analysis
 set values above anticipated solution to avoid initial cell collapse from applied positive pressure
 pcellmin and pcellmax set limits to solution range

- nb inflation pressure pinfl() defined earlier is an additional applied constant loading and separate from gas cell internal pressure/volume controls

DATA

- nflr
nr norient gamma zhead [1,nflr]

DESCRIPTION

CONSTANT HEAD FLUID LOADING

nflr no. of fluid loaded regions
nr region no.
norient norient = 1 fluid load on membrane top surface
norient = -1 bottom ..
gamma fluid specific weight (kN/m3)[lbf/ft3]
zhead level of fluid constant head (m)[ft]

Release Notes

- 3.23** External Pressure Loading
- 3.24** Air Cells converted to Gas Cells
- 3.25** Panel Defined Snow Loading
- 3.26** File Descriptors
Node Restraint Modifications

Details

Cubic spline interpolated factors over panel zones added
Previous LD files with wind cases defined must be updated

Unit Lift entered to model differential gas pressure (eg helium inside cell, air outside)
Differential pressure is zero at reference elevation
Total pressure at height Zp : Pcell + Unit_Lift x (Zp-Zcell)

Partial panel zones defined by local triangle nos
Multiple zones may be applied to individual panels
Previous LD files with panel defined snow must be updated

First two lines of file must be text which is automatically assigned as the LR run description text

Setting nrsl negative releases system line internal nodes only

DATA**DESCRIPTION****Release Notes****Details**

3.27	Metric / Imperial Units	Imperial Units set by entering 'Imperial' on line 4 of PROJECT file Metric Units by entering 'Metric' on line 4, or line 4 not defined
3.28	Slip Cable Specified Tensions	Specified slip cable tensions via nscten variable
3.29	Air Cell Min/Max Pressures	Estimated limits to cell final internal pressure. Aid convergence, but set also set limits to possible solution. If equilibrium pressure is outside this range, then cells will not converge to correct solution
3.30	Air Cell Reference Volumes	Air cell reference volumes usually calculated from ANALYS start geometry. Option added to permit entry of specified volume. Reference volumes are associated with defined cell reference pressure pcell.

Panel Orientation and Compensation Data may be defined in a **CP file** for use with **CUTPAT 3.19** and above.

A separate CP file is needed for each region of the structure. The main benefits of using the CP file rather than manual compensation input are:

- Single and sheared panels may be rotated about reference orientation axis
- Varying orientations and compensations within a region can be entered once and the whole region patterned in a single sequence
- Compensation applied to sheared panels using values from first panel of pair (ie bilinear warp compensation handled automatically)
- Decompensation held over specified length and choice of blend type only accessible via CD or CP file
- Intermediate warp and fill compensation values may be defined along panel for linear interpolation between successive values

DATA	DESCRIPTION
nr	nr region number
nop	nop no of output panels defined
np1 np2 [1,nop]	np1 np2 panel nos (np2=0 if single panel)
ipx {if ipx=4} nx1 nx2	ipx output panel reference axis 1 = side 1 2 = side 2 3 = common (sheared) mean (single) 4 = through nodes nx1 to nx2
pxtheta	pxtheta panel rotation about ref axis (deg (+)ve clockwise)
nvp pl p2 [1,nvp]	nvp no of panel compensation sequences definition p1 p2 sequence start and finish local panel nos
w11 w12 w21 w22	w11 w12 side one end 1 & end 2 warp % compensation w21 w22 side two
f1 f2	f1 f2 end 1 & end 2 fill % compensation
nicd {if nicd > 0}	* nicd no of sets of intermediate compensation data
dli w1i w2i fi [1,nicd]	dli relative length along panel (between 0.0 and 1.0) w1i w2i fi warp side 1, warp side 2 and fill % compensation
df1 dapp11 dapp21 dlen1 idcb1	df1 end 1 fill % decompensation
df2 dapp12 dapp22 dlen2 idcb2	dapp11 end 1 side 1 length for constant decompensation dapp21 end 1 side 2 dlen1 end 1 length of blend from df1 to full compensation idcb1 end 1 blend type 0 = quadratic 1 = linear df2 dapp12 ... end 2 decompensation variables

Region Compensation Data may be defined in a **CD file** for use with **CUTPAT 3.17** and above.

A separate CD file is needed for each region of the structure. The main benefits of using the CD file rather than manual compensation input are:

- Varying compensations within a region can be entered once and the whole region patterned in a single sequence
- Enables correct assignment of bilinear warp compensations across sheared panels
- Decompensation held over specified length and choice of blend type only accessible via CD file

Varying warp and fill compensations are assigned according to the total length and width range of each individual panel, even if panels are subsequently sheared together.

DATA	DESCRIPTION
• nr	nr region number
nvp	nvp no of panel sequences defined
pl p2 [1,nvp]	p1 p2 sequence start and finish local panel nos
w11 w12 w21 w22	w11 w12 side one end 1 & end 2 warp % compensation
f1 f2	w21 w22 side two f1 f2 end 1 & end 2 fill % compensation
df1 dapp11 dapp21 dlen1 idcb1	df1 end 1 fill % decompensation
df2 dapp12 dapp22 dlen2 idcb2	dapp11 end 1 side 1 length for constant decompensation
	dapp21 end 1 side 2
	dlen1 end 1 length of blend from df1 to full compensation
	idcb1 end 1 blend type 0 = quadratic 1 = linear
	df2 dapp12 ... end 2 decompensation variables

MEMPLOT uses the standard inTENS **MPLLOT** graphic display routines to provide visualisation of TG FG & LR data files. The same routines and user interface are used for run-time graphic display and output within TOPGEN, GENFORM and ANALYS. A resizable window shows an image as it would appear on the current sheet size when printed. Printing is via the standard Windows printer driver to any suitable connected device. Detailed plot layout and sheet size options are available to the user through the MPLLOT.CON configuration file.

The model elements on display and the view are controlled via the keyboard. Model structural elements are toggled on and off as required by repeated pressing of the relevant key. Normally a lower-case entry displays the full element and an upper-case entry displays shrunken elements. The view is adjusted by moving an Observer around the Model.

MODEL DATA	Key	Element
	[x]	AXES for view orientation (Red(X) Green(Y) Blue(Z))
	[y]	SYSTEM LINES
	[r]	RIDGE LINES
	[w]	WARP LINES
	[n] [N]	ATTACHED NODES (Current / All)
	[CTRL][n]	SYSTEM NODES
	[m] [M]	MEMBRANE ELEMENTS
	[CTRL][m]	MEMBRANE STRESSES (kN/m)[pli]
	[l] [L]	LINKS
	[CTRL][l]	LINK TENSIONS (kN)[kip]
	[c] [C]	SLIP CABLES
	[CTRL][c]	SLIP CABLE TENSIONS (kN)[kip]
	[b] [B]	BEAMS
	[s] [S]	SPACER STRINGS

MODEL DATA**Key****Element**

[v]	EXTERNAL PRESSURE LOADING (kN/m2)[psf]
[V]	SNOW LOADING (kN/m2)[psf]
[h] [H]	CONTOURS (Default / User Defined)
[INSERT]	DISPLAY ALL ELEMENTS (except Membranes)

VIEW CONTROL**Key****Function**

[←] [↑] [↓] [→]	[ARROW] Keys used for moving Observer relative to Model :
[ARROW]	Rotate Observer in Arrow Direction
[SHIFT][ARROW]	Fine Rotate Observer
[ALT][ARROW]	Translate Observer in Arrow Direction
[CTRL][ARROW]	Fine Translate Observer
[p]	Full XY-Plan View
[e]	Full XZ-Elevation
[E]	Full YZ-Elevation
[z]	Zoom via Cursor-Defined Window
[Z]	Zoom In by 10%
[CTRL][z]	Zoom Out by 10%
[f]	Zoom Out to Full View
[HOME]	Centre View on Cursor-Defined Point

ADDITIONAL CONTROLS	Key	Function
	[DEL]	Clear Current View
	[CTRL][p]	Print Current View via Windows Printer Dialogue
	[CTRL][d]	Print Current View to Default Printer
	[CTRL][b]	Output 'image.bmp' File
	[CTRL][t]	Output 'TP_<sourcefile>.dat' Tecplot Format File (MEMPLOT only)
	[q] [Q]	Quit Graphic Display
	[F1]	Edit Current Plot Scale
	[F2]	Select Current Plotter and Sheet Size
	[F3]	Direct View Angle Definition
	[F4]	Select Active Membrane Regions
	[SHIFT][F4]	Select Active Gas Cells
	[CTRL][F4]	Select Active Element Groups
	[F5]	Toggle Warp/Fill Membrane Stress Display
		Toggle Cpe/Pe/Qe External Pressure Loading Display
		Toggle Ks/S/So Snow Loading Display
	[SHIFT][F5]	Toggle Membrane Borders for Stress & Loading Display
	[F6]	Toggle Averaged/Element Membrane Stress Display (MEMPLOT + LR only)
	[F7]	Select Stress/Loading Display Variable Range
	[F11]	Toggle Colour/Monochrome Element Display

	[F12]	Toggle White/Black Background Colour
Release Notes	4.02	Details
	[n] [N]	Keys reversed for [Current] / [All Attached] Nodes
	[y]	System Line Display extended to include Ridge and Passive SL's
	[CTRL][d]	Print Current View to Windows Default Printer without Dialogue
	[F1]	Current View Centre maintained when Scale Adjusted
	[F3]	Direct Entry of View Angles
	Printing	Black Background automatically switched to White for Printing Axes switched off during Printing Node Numbering Character Size Prints Correctly
	MPLLOT.CON	Additional Printer Data Defined : Page Size and Orientation set automatically in Printer Dialogue Image auto set Monochrome for Monochrome Defined Printers User Adjustable Screen & Plotter Node/Label Character Sizes User Defined Default Screen Background Colour (Black/White

Release Notes		Details
	4.03	
	[F6]	Toggles Averaged/Element Membrane Stresses for MEMPLOT + LR only All other MPLOT applications show element stresses (now current in ANALYS)
	[CTRL][b]	Output Current View to 'image.bmp' File
	4.04	
	MPLOT.CON	User Defined Logo option added
	[v]	Displays External Pressure Cpe/Pe/Qe toggled via [F5] (in ANALYS only)
	4.05	
	[V]	Displays Snow Loading Ks/S/So toggled via [F5] (in ANALYS only)
	[F7]	Enables user defined variable range for Stress/Loading displays
	Printing	Revised greyscale display/printing of Stress/Loading maps in monochrome
	4.06	
	[CTRL][t]	Output 'TP_<sourcefile>.dat' Tecplot Format File (MEMPLOT only)
	[v] [V]	Now available in ANALYS and MEMPLOT+LR
	[h]	Contours all membrane with default Z increment (default set in MPLOT.CON)
	[H]	User defined contour range and Z increment
	[SHIFT][F4]	Select active gas cell membrane regions
	[CTRL][F4]	Select active elements by group number

Release Notes

4.07

Details

Metric / Imperial Unit option included

4.08

[CTRL][I]

Display Link Tensions

4.09

[CTRL][c]

Display Slip Cable Tensions

4.10

[SHIFT][F5]

Toggle membrane element borders for stress and loading display

4.11

[s] [S]

Display spacer strings

System Line Data may be defined in a **SL file** for use with **GEOMETRY 3.04.0** and above.

Option 3.10 of GEOMETRY solves for the intercept of warp lines with the vertical projection of system lines defined in an SL file. It calculates the length along the warp lines to intercept, and is primarily intended for use with TOPMOD.

The system lines are defined as a sequence of point data X and Y coordinates. The system line shape may be defined as either polygonal or by a cubic spline curve fit through the data points.

DATA	DESCRIPTION
<ul style="list-style-type: none"> • nns1 islmod 	nns1 no of data points defining system line islmod system line shape mode islmod = 0 polygonal through data points 1 cubic spline through data points
x y [1,nns1]	x y data point plan coordinates

Point Coordinate Data may be defined in an **XD file** for use with **GEOMETRY 3.04.0** and above.

Option 3.6 of GEOMETRY solves for the membrane surface intercept of arbitrary vectors through given data points. The calculated points may be output into an SP file for subsequent markup of the points in CUTPAT.

The original data points may be entered individually or through an XD file. The data point reference number is independent of the structure node numbering scheme.

DATA	DESCRIPTION
• ndp	ndp no of data points
• nref x y z [1,ndp]	nref data point reference number x y z data point plan coordinates

DATA

- nwlmg
ir iws iwf cl1s cl2s cl1f cl2f [1,nwlmg]
- nslmps
ir ip ipside isl cl1 cl2 [1,nslmps]
- npsin
ir ip iend npin [1,npsin]
nref x y z [1,npin]

DESCRIPTION

CROSS PANEL CUTS

nwlmg	no. of warp line modification groups
ir	region no.
iws	start warp line no.
iwf	finish warp line no.
cl1s	start warp line : end 1 correction length
cl2s	end 2
cl1f	finish warp line : end 1 correction length
cl2f	end 2
nslmps	no. of system line modified panel sides
ir	region no.
ip	panel no.
ipside	panel side no. ... 1 or 2
isl	system line to be modified
cl1 cl2	system line : end 1 end 2 correction length
npsin	no of panels with specified intermediate nodes for cross panel cuts defined above
ir	region no.
ip	panel no.
iend	panel end ... 1 or 2
npin	no of intermediate nodes
nref	intermediate node reference no.
x y z	intermediate node co-ordinates

Unless npsin specified intermediate nodes are utilised TOPMOD will generate a straight cut across the panel with intermediate nodes interpolated onto the surface as necessary. TOPMOD will check that the intermediate node co-ordinates defined for npsin do lie on the membrane surface.

DATA

- nemp
ir ip ipside imode
isl1 cl1 isl2 cl2 npin [1,nemp]
nref x y z [1,npin]

- npmerge
ir ip1 [1,npmerge]

DESCRIPTION

PANEL SIDE MODIFICATIONS

nemp	no. of single edge modified panels
ir	region no.
ip	panel no
ipside	panel side ... 1 or 2
imode	cut mode ... 1 discard cut & retain panel 2 retain cut & discard panel
isl1 cl1	cut end 1 ... system line and length along
isl2 cl2	cut end 2
npin	no. of intermediate nodes on cut line
nref	intermediate node reference no.
x y z	intermediate node co-ordinates

PANEL MERGE

npmerge	no of panel pairs to be merged
ir	region no.
ip1	panel no. of first panel in pair to be merged

TOPMOD used to create reduced membrane topologies for reinforcement and lining membranes from main structure form

Input TG FG and TM (topology modification) files

Output TGM and FGM modified TG and FG files which can be used in MEMPLOT and CUTPAT (nb these are not an equilibrium form)

Reduced panels are defined by setting new limits along warp and system lines:

cl1 - distance from warp/SL end 1 to start of reduced panel	(set 0.000 if no end 1 reduction)
cl2 - finish	(set 0.000 if no end 2 reduction)

(negative values for cl1/cl2 indicate dimension from warp/SL end 2
values are interpolated linearly between start and finish warp lines in a definition group)