

Element Reference Manual

Version 15.0 Issue 1

LUSAS

Forge House, 66 High Street, Kingston upon Thames,
Surrey, KT1 1HN, United Kingdom

Tel: +44 (0)20 8541 1999

Fax +44 (0)20 8549 9399

Email: info@lusas.com

<http://www.lusas.com>

Distributors Worldwide

Copyright ©1982-2014 LUSAS

All Rights Reserved.

Table of Contents

Notation.....	1
Overview.....	7
Element selection.....	7
Element uses.....	7
Element Groups.....	11
Element Sub-Groups.....	11
Element Types and Availability.....	12
Element Index.....	15
Element Summary Tables.....	39
Chapter 1 : Bar Elements.....	57
2D Structural Bar Elements.....	59
3D Structural Bar Elements.....	64
Chapter 2 : Beam Elements.....	69
2D Engineering Thick Beam Element.....	71
3D Engineering Thick Beam Element.....	76
2D Engineering Grillage Thick Beam Element.....	82
3D Thick Beam Element (nonlinear).....	86
3D Thick Beam Elements.....	93
3D Thick Beam Elements with Quadrilateral Cross-Section.....	101
2D Kirchhoff Thin Beam Elements.....	108
2D Kirchhoff Thin Beam Element with Quadrilateral Cross-Section.....	114
3D Kirchhoff Thin Beam Elements.....	121
3D Kirchhoff Thin Beam Element with Quadrilateral Cross-Section.....	127
3D Semiloof Thin Beam Elements.....	134
3D Semiloof Thin Beam Element with Quadrilateral Cross-Section.....	140
Chapter 3 : 2D Continuum Elements.....	147
2D Plane Stress Continuum Elements.....	149
2D Plane Stress Continuum Element with Enhanced Strains.....	155
2D Plane Stress Continuum Crack Tip Elements.....	161
2D Plane Stress Explicit Dynamics Elements.....	166
2D Plane Strain Continuum Elements.....	171
2D Plane Strain Continuum Element with Enhanced Strains.....	177
2D Plane Strain Continuum Element for Large Strains.....	183
2D Plane Strain Continuum Crack Tip Elements.....	187
2D Plane Strain Explicit Dynamics Elements.....	193
2D Plane Strain Two Phase Continuum Elements.....	198
2D Axisymmetric Solid Continuum Elements.....	203
2D Axisymmetric Solid Continuum Element with Enhanced Strains.....	209
2D Axisymmetric Solid Continuum Element for Large Strains.....	215
2D Axisymmetric Solid Continuum Crack Tip Elements.....	220
2D Axisymmetric Solid Explicit Dynamics Elements.....	226
2D Axisymmetric Solid Two Phase Continuum Elements.....	231
2D Axisymmetric Fourier Ring Elements.....	237
Chapter 4 : 3D Continuum Elements.....	243
3D Solid Continuum Elements.....	245
3D Solid Continuum Element with Enhanced Strains.....	252
3D Solid Continuum Crack Tip Elements.....	258
3D Solid Continuum Composite Elements (Tetrahedral).....	264
3D Solid Continuum Composite Elements (Pentahedral and Hexahedral).....	270
3D Solid Continuum Explicit Dynamics Elements.....	276
3D Solid Two Phase Continuum Elements.....	281
Chapter 5 : Plate Elements.....	287
2D Isoflex Thin Plate Flexure Elements.....	289

2D Isoplex Thick Plate Flexure Element	293
2D Mindlin Thick Plate Flexure Element	297
Chapter 6 : Shell Elements.....	303
2D Axisymmetric Thin Shell Element.....	305
3D Flat Thin Shell Elements.....	311
3D Flat Thin Nonlinear Shell Element	317
Semiloof Curved Thin Shell Elements	323
3D Thick Shell Elements	331
Chapter 7 : Membrane Elements.....	341
2D Axisymmetric Membrane Elements	343
3D Space Membrane Elements.....	348
Chapter 8 : Joint Elements.....	353
2D Joint Element for Bars, Plane Stress and Plane Strain	355
2D Joint Element for Engineering and Kirchhoff Beams	359
2D Joint Element for Grillage Beams and Plates	364
2D Joint Element for Axisymmetric Solids.....	368
2D Joint Element for Axisymmetric Shells	372
3D Joints for Bars, Solids and Space Membranes	376
3D Joints for Semiloof Shells	380
3D Joint Elements for Engineering, Kirchhoff and Semiloof Beams	384
3D Joint Element for Semiloof Beams	388
Chapter 9 : Thermal / Field Elements.	393
2D Bar Field Elements.....	395
2D Axisymmetric Membrane Field Elements	399
3D Bar Field Elements.....	403
2D Link Field Element	407
3D Link Field Element	410
2D Axisymmetric Link Field Element	413
2D Plane Field Elements	416
3D Solid Field Elements.....	420
3D Solid Composite Field Element (Tetrahedral)	425
3D Solid Composite Field Elements (Pentahedral and Hexahedral)	430
2D Axisymmetric Field Elements	435
Chapter 10 : Interface Elements.....	441
2D Interface Element.....	443
3D Interface Element.....	447
Chapter 11 : Non-Structural Mass Elements.....	451
2D Point Mass Element	453
3D Point Mass Element	456
3D Line Mass Elements.....	459
2D Line Mass Elements.....	462
Surface Mass Elements	465
Chapter 12 : Rigid Elements.	469
Rigid Surface 2D Elements	471
Rigid Surface 3D Elements	474
Appendix A : Element and Pressure Loads.	477
ELDS Element Loads	477
ENVIT/DET Environmental Temperature Loading	480
FLD Face loading applied to thermal bars.....	481
Face Loads On 2D Continuum Elements.....	482
Face Loads On 3D Continuum Elements	483
UDL Loads on Shells.....	485
Appendix B : Element Restrictions.....	487
Mid-side Node Centrality	487
Excessive Element Curvature	487
Excessive Aspect Ratios	487
Excessive Warping.....	488

Appendix C : Local Element Axes	489
Standard Joint Element	489
Standard Line Element	489
Standard Surface Element.....	489
Appendix D : Sign Conventions.	491
Standard Bar Element.....	491
Standard Beam Element	491
Grillage Elements.....	492
2D Engineering Beam Elements.....	493
3D Engineering Beam Elements.....	494
Standard Beam Eccentricities.....	496
Standard 2D Continuum Element.....	496
Standard 3D Continuum Element.....	496
Standard Plate Element	497
Thin Shell Element.....	497
Thin Shell Eccentricity.....	498
Thick Shell Element	499
Thick Shell Eccentricity	499
Standard Membrane Element	500
Standard Field Element	500
Standard Joint Element	500
Appendix E : Thick Shell Notation.....	501
Thick Shell Nodal Rotation	501
Appendix F : Newton Coates Integration.	505
Newton-Cotes Integration Points	505
Appendix G : Shear Area and Torsional Constant.	507
Shear Areas	507
Torsional Constant	508
Appendix H : Principal Stress Output.	511
Output Notation for Principal Stresses.....	511
Appendix I : Mass Lumping.	513
Mass Lumping in LUSAS	513
Appendix J : Moments of Inertia.....	515
Moments of Inertia Definitions	515
Appendix K : Results Tables.	517
Key to Element Results Tables	517
Transforming Results Directions	525
2D Structural Bars BAR2, BAR3.....	527
3D Structural Bars BRS2, BRS3.....	528
2D Engineering Beam BEAM.....	529
3D Engineering Thick Beam BMS3	530
2D Engineering Grillage Thick Beam GRIL.....	531
3D Thick Beam (Nonlinear) BTS3.....	532
3D Thick Beam Elements BMI21, BMI22, BMI31, BMI33, BMX21, BMX22, BMX31, BMX33.....	533
2D Kirchhoff Thin Beams BM3, BMX3	534
3D Kirchhoff Thin Beams BS3, BS4, BSX4.....	535
3D Semiloof Thin Beams BSL3, BSL4, BXL4	536
2D Continuum (Plane Stress) TPM3/6, QPM4/8, QPM4M, TPK6, QPK8.....	537
2D Continuum Plane Stress (Explicit Dynamics) TPM3E, QPM4E	538
2D Continuum (Plane Strain) TPN3/6, QPN4/8, TNK6, QNK8, QPN4M	539
2D Continuum (Plane Strain) QPN4L	540
2D Plain Strain Two Phase Continuum TPN6P, QPN8P.....	541
2D Continuum Plane Strain (Explicit Dynamics) TPN3E, QPN4E.....	542
2D Continuum Axisymmetric Solid (Explicit Dynamics) TAX3E, QAX4E	543
2D Axisymmetric Solid Two Phase Continuum TAX6P, QAX8P	544
2D Continuum Axisymmetric Solid Fourier TAX3/6F, QAX4/8F	545

Axisymmetric Solid TAX3/6, QAX4/8, QAX4M, TXK6, QXK8	546
Axisymmetric Solid Large Strain QAX4L	547
3D Solid Continuum TH4/10, TH10S, PN6/12/15, PN6L/12L, HX8/16/20, HX8M, HX8L/16L, TH10K, PN15K, HX20K	548
3D Solid Continuum Two Phase TH10P, PN12P, PN15P, HX16P, HX20P	549
3D Solid Continuum Explicit Dynamics TH4E, PN6E, HX8E	550
Isoflex Thin Plates TF3, QF4	551
Isoflex Thick Plates QSC4	552
Mindlin Thick Plates TTF6, QTF8	553
2D Axisymmetric Membranes BXM2, BXM3	554
3D Space Membranes TSM3, SMI4	555
Axisymmetric Shells BXS3	556
3D Flat Thin Shells TS3, QSI4	557
3D Flat Thin Nonlinear Shell TSR6	558
Semiloof Shells TSL6, QSL8	559
Thick Shells TTS3, TTS6, QTS4, QTS8	560
2D Joints (for Bars, Plane Stress and Plane Strain) JNT3	561
2D Joints (for Engineering and Kirchhoff Beams) JPH3	562
2D Joints (for Grillage Beams and Plates) JF3	563
2D Joints (for Axisymmetric Solids) JAX3	564
2D Joints (for Axisymmetric Shells) JXS3	565
3D Joints (for general 3 dof connection) JNT4, JL43	566
3D Joints (for general 6 dof connection) JSH4, JL46	567
3D Joints (for Semiloof Element Mid-side Nodes) JSL4	568
Thermal Bars BFD2/3, BFS2/3, BFX2/3	569
Thermal Links LFD2, LFS2, LFX2	569
Plane and Axisymmetric Field TFD3/6, QFD4/8, TFX3/6, QXF4/8	569
Solid Field TF4/10, PF6/12/15, HF8/16/20, TF10S, PF6C/12C, HF8C/16C	570
2D Interface Element IPN4, IPN6, IAX4, IAX6	571
3D Interface Element IS6, IS8, IS12, IS16	572
Appendix L : Joint Element Compatibility	573
Joint Element Compatibility	573
Index	575

Notation.

A	Cross sectional area
A_p	Plastic area
A_s, A_{sy}, A_{sz}	Effective shear area
A₁ ... A_n	Nodal cross sectional areas
ar	Mass Rayleigh damping constant
α	Coefficient of thermal expansion
α_s	Softening parameter
$\alpha_x, \alpha_y, \alpha_z, \alpha_{xy}, \alpha_{xz}, \alpha_{yz}$	Orthotropic thermal expansion coefficients
$\alpha_x, \alpha_y, \alpha_z$	Angular accelerations
br	Stiffness Rayleigh damping parameter
β	Shear retention factor/parameter
β	Principal stresses direction
C	Specific heat capacity
C_i	(i)th hardening stiffness
C₀	Neo-Hookean rubber model constant
C₁, C₂	Mooney-Rivlin rubber model constants
c	Cohesion
co	Initial cohesion
D_{ij}	Rigidity coefficients
du, dq	Relative displacement, rotation
E	Modulus of elasticity (Young's modulus)

Ep	Elasto-plastic modulus
Ex, Ey, Ez	Orthotropic moduli of elasticity
ep	Strain at peak compressive strength
ey, ez	Eccentricity
$\epsilon_x, \epsilon_y, \epsilon_z$	Direct strains (local or global)
ϵ_s	Maximum shear strain
ϵ_e	Von Mises equivalent strain
ϵ_c	Creep strains
ϵ_p	Equivalent plastic strain
Fx, Fy, Fz	Forces (local or global)
Fyld	Yield force
F	Deformation gradient
fc'	Compressive strength of concrete
ft'	Tensile strength of concrete
ψ_x, ψ_y, ψ_z	Flexural (bending) strain resultants
$\psi_{xy}, \psi_{xz}, \psi_{yz}$	Torsional strain resultants
G	Shear modulus
Gf	Fracture energy
Gxy, Gxz, Gyz	Orthotropic shear moduli
$\gamma_x, \gamma_y, \gamma_z$	Membrane strain resultants
$\gamma_x, \gamma_y, \gamma_z$	Field gradients (local or global)
H	Enthalpy
H_i	Isotropic hardening parameter
H_k	Kinematic hardening parameter
hc	Convective heat transfer coefficient
hf	Heat fraction
hr	Radiative heat transfer coefficient

$\theta_{\mathbf{x}}, \theta_{\mathbf{y}}, \theta_{\mathbf{z}}$	Rotations (local or global)
θ_1, θ_2	Loof node rotations (local)
$\theta_{\alpha}, \theta_{\beta}$	Nodal rotations for thick shells
θ_{λ}	Angle defining principal directions of λ_1, λ_2
$\mathbf{I}_{\mathbf{y}}, \mathbf{I}_{\mathbf{z}}$	1st moments of inertia
$\mathbf{I}_{\mathbf{yy}}, \mathbf{I}_{\mathbf{zz}}$	2nd moments of inertia
$\mathbf{I}_{\mathbf{yz}}$	Product moment of inertia
$\mathbf{J}$	Volume ratio (determinant of $\mathbf{F}$)
$\mathbf{K}$	Spring stiffness
$\mathbf{K}_{\mathbf{c}}$	Contact stiffness
$\mathbf{K}_{\mathbf{l}}$	Lift-off stiffness
$\mathbf{K}_{\mathbf{o}}$	Original gap conductance
$\mathbf{K}_{\mathbf{t}}$	Torsional constant
$\mathbf{k}$	Thermal conductivity
$\mathbf{k}_{\mathbf{x}}, \mathbf{k}_{\mathbf{y}}, \mathbf{k}_{\mathbf{z}}$	Orthotropic thermal conductivities
$\mathbf{k}_{\mathbf{r}}$	Bulk modulus
κ	Hardening stiffness
$\mathbf{L}_{\mathbf{i}}$	Limit of (i)th hardening stiffness
$\lambda_1, \lambda_2, \lambda_3$	Principal stretches
$\mathbf{M}$	Mass
$\mathbf{M}_{\mathbf{x}}, \mathbf{M}_{\mathbf{y}}, \mathbf{M}_{\mathbf{z}}$	Concentrated moments (local or global)
$\mathbf{M}_{\mathbf{x}}, \mathbf{M}_{\mathbf{y}}, \mathbf{M}_{\mathbf{z}}, \mathbf{M}_{\theta}$	Flexural moments (local or global)
$\mathbf{M}_{\mathbf{xy}}, \mathbf{M}_{\mathbf{xz}}, \mathbf{M}_{\mathbf{yz}}$	Torsional moments (local or global)
$\mathbf{M}_1, \mathbf{M}_2$	Concentrated loof moments (local or global)
$\mathbf{m}_{\mathbf{x}}, \mathbf{m}_{\mathbf{y}}, \mathbf{m}_{\mathbf{z}}$	Mass in element local directions
μ	Coulomb friction coefficient
$\mu_{\mathbf{ri}}, \alpha_{\mathbf{ri}}$	Ogden rubber model constants

Nx, Ny, Nz, N_θ Membrane resultants (local or global)

Nx, Ny, Nxy Stress resultants

Nmax, Nmin Principal stress resultants

Ns Maximum shear stress resultant

Ne Von Mises equivalent stress resultant

ν Poisson's ratio

ν_{xy}, ν_{xz}, ν_{yz} Orthotropic Poisson's ratio

Px, Py, Pz Concentrated loads (global)

ρ Mass density

Q Field loading

qa Field face loading flux/unit area

qv Field volume loading flux/unit volume

qx, qy, qz Field fluxes (local or global)

qs Stress potential parameters

Sp Plastic shear area

σ_y Yield stress

σ_{y0} Initial uniaxial yield stress

σx, σy, σz Direct stresses (local or global)

σmax, σmin Principal stresses

σxy, σxz, σyz Shear stresses (local or global)

σs Maximum shear stress

σe Von Mises equivalent stress

T Temperature

T, T₀ Final, initial temperatures

t₁ ... t_n Nodal thicknesses

U, V, W Displacements (global)

Φ Field variable

Φ_e	External environmental temperature
ϕ	Frictional angle
ϕ_0	Initial frictional angle
ϕ	Body force potential
V_x, V_y, V_z	Nodal velocities (global)
$V_{11}, V_{12} \dots V_{33}$	Left stretch tensor components
W_x, W_y, W_z	Uniformly distributed intensities
X, Y, Z	Nodal coordinates (global)
$X_{cbf}, Y_{cbf}, Z_{cbf}$	Constant body forces (global)
X_o, Y_o, Z_o	Offsets of finite element model coordinate system from point about which global angular acceleration and velocities are applied
$y_1, z_1 \dots y_4, z_4$	Cross sectional coordinates (local)
Z_{yp}, Z_{zp}	Torsional plastic moduli
Z_{yyp}, Z_{zzp}	Flexural plastic moduli
ω	Frequency of vibration
$\Omega_x, \Omega_y, \Omega_z$	Angular velocities (global)

Introduction

Overview

The *Element Reference Manual* describes the elements currently available in LUSAS. It has been designed to be used in conjunction with the *Solver Reference Manual* and provides input/output information which is specific to each element type.

If you require:

- [General](#) theoretical information - refer to *Theory Manual Volume 1*
- [Element related](#) theoretical / formulation information - refer to *Theory Manual Volume 2*

Element selection

Details of typical [element uses](#) are provided and, to assist you with choosing an element for a particular modelling task, three alternative selection methods are available for selecting by:

- ❑ [Element type](#) - listing just element group, sub-group and element name
- ❑ [Element index](#) - showing element name, geometry, nodal freedoms and element availability
- ❑ [Element summary](#) - showing element names, material property, loading, nonlinear, integration, and mass modelling capabilities

Of these three methods, the element summary tables provide the most detail to enable correct element selection for a particular modelling task.

Element uses

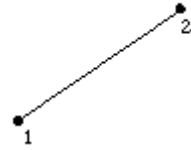
The following brief descriptions of each element group are provided to assist you with element selection for a particular modelling task.

Additional more detailed and element-specific recommendations on use can be found by viewing the Recommendations on Use section provided within each element's listing. For an example see [3D Isoparametric Thick Beam Elements](#)

Bar Elements

Bar elements are used to model plane and space truss structures, cables in cable-stayed structures, and stiffening reinforcement.

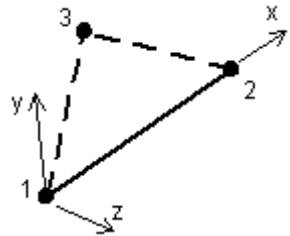
- LUSAS incorporates 2 and 3-dimensional bar elements which may either be straight or curved.
- Bar elements model **axial force** only.



Beam Elements

Beam elements are used to model plane frames, space frame structures, and cables in cable-stayed structures.

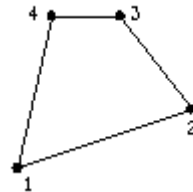
- LUSAS incorporates a variety of thin and thick beams in both 2 and 3-dimensions. In addition, specialised beam elements for modelling grillage or eccentrically ribbed plate structures are available.
- LUSAS beam elements may be either straight or curved and may model **axial force**, **bending** and **torsional behaviour**.



2D Continuum Elements

2D continuum elements are used to model solid structures whose behaviour **may reasonably be assumed to be 2-dimensional**.

- 2D continuum elements may be applied to plane stress, plane strain and axisymmetric solid problems.
- Triangular and quadrilateral elements are available.
- Fourier elements, which allow non-axisymmetric loading to be applied to axisymmetric models, are considered a special case of the 2D continuum elements since the mesh is defined entirely in the xy-plane, but the resulting displacements, strains and stresses are fully three-dimensional.
- Special crack tip elements are available to model the singularities encountered at crack opening
- Explicit elements are available to model high speed dynamics problems efficiently.



3D Continuum Elements

3D continuum elements are used to model *fully3-dimensional structures*.

- Tetrahedral, pentahedral and hexahedral solid elements are available to model full 3-dimensional stress fields.
- Composites elements are available to model laminates.
- Special crack tip elements are available to model the singularities encountered at crack opening

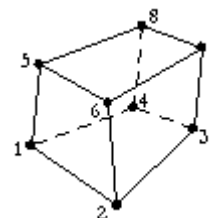
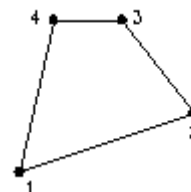


Plate Elements

Plate elements are used to model flat structures whose deformation can be assumed to be predominantly flexural

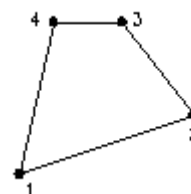
- LUSAS incorporates both thin and thick plate elements.
- Triangular and quadrilateral flexural plate elements are available.



Shell Elements

Shell elements are used to model 3-dimensional structures whose behaviour is dependent upon both *flexural and membrane effects*.

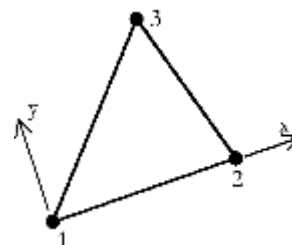
- LUSAS incorporates both flat and curved shell elements.
- Triangular and quadrilateral elements are available
- Both thin and thick shell elements are available.



Membrane Elements

Membrane elements are used to model 2 and 3-dimensional structures whose behaviour is dominated by in-plane membrane effects.

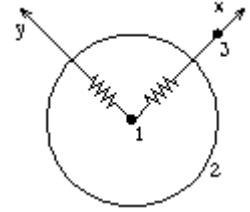
- LUSAS incorporates both axisymmetric and space (3-dimensional) membrane elements.
- Membrane elements incorporate *in-plane (membrane) behaviour only* (they include no bending behaviour).



Joint Elements

Joint elements are used to model *flexible joints* between other LUSAS elements.

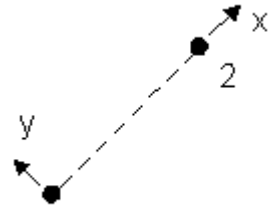
- LUSAS incorporates a variety of joint elements which are designed to match the nodal freedoms of their associated elements.
- Joint elements may also be used to model point masses, elasto-plastic hinges, or smooth and frictional element contacts.



Non-Structural Mass Elements

Non-Structural Mass elements are used to model translational *mass* at a point, along an edge or on a surface.

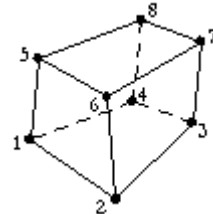
- Non-Structural Mass elements must be used with other structural elements.



Field Elements

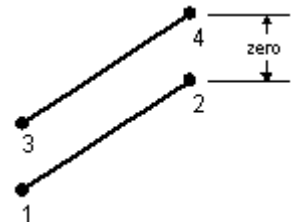
Field elements are used to model quasi-harmonic equation problems such as thermal conduction or potential distribution.

- LUSAS incorporates bar, plane, axisymmetric solid and 3-dimensional solid field elements.
- Thermal link elements are also available.



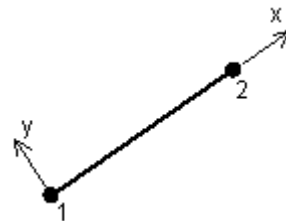
Interface Elements

Interface elements should be used at places of potential delamination between 2D continuum elements for modelling *delamination* and *crack propagation*.



Rigid Elements

Rigid elements are used to define the shape of a rigid surface which is not part of the analysis model.



Element Groups

The LUSAS Element Library is arranged into the following element groups:

- ☐ **Bars**
- ☐ **Beams**
- ☐ **2D Continuum elements**
- ☐ **3D Continuum elements**
- ☐ **Plates**
- ☐ **Shells**
- ☐ **Membranes**
- ☐ **Joints**
- ☐ **Non-structural mass elements**
- ☐ **Thermal/Field elements**
- ☐ **Interface elements**
- ☐ **Rigid elements**

Element Sub-Groups

Each element group is also sub-divided into element sub-groups according to the type of element formulation as shown in the following [table](#). For example, the **Beam** element group contains the element sub-groups: Engineering beams, Thick beams, Kirchhoff beams and Semiloof beams.

Within each sub-group elements vary according to the geometry, the number of nodes, and the properties required by each element. The individual elements are referred to by their LUSAS name. For example, the **Engineering beams** are named BEAM, BMS3 and GRIL.

Note

The dimensional classification of LUSAS elements is on the basis of the number of dimensions required for input of the nodal coordinates. For example, an engineering grillage element, (GRIL) requires X, Y coordinates and is hence classed as being 2-dimensional (despite having an out of plane displacement freedom).

Element Types and Availability

Element Group	Element Subgroup	Element Name and Software Product Version Availability		
		LT	Standard	Plus
Bars	Structural bars	BAR2 , BRS2	BAR3 , BRS3	
Beams	Engineering beams	BEAM , BMS3 , GRIL		
	Thick beams	BMI21	BTS3 , BMX21	BMI22 , BMX22 , BMI31 , BMX31 , BMI33 , BMX33
	Thin (Kirchhoff) beams		BM3 , BMX3	BS3 , BS4 , BSX4
	Semiloof beams			BSL3 , BSL4 , BXL4
2D Continuum	Plane stress continuum		TPM3 , TPM6 , OPM4 , OPM8 , OPM4M , TPK6 , QPK8	TPM3E , OPM4E
	Plane strain continuum		TPN3 , TPN6 , QPN4 , QPN8 , QPN4M , QPN4L , TNK6 , QNK8	TPN3E , QPN4E
	Plain strain two phase		TPN6P , QPN8P	
	Axisymmetric solid continuum		TAX3 , TAX6 , QAX4 , QAX8 , QAX4M , QAX4L , TXK6 , QXK8	TAX3E , QAX4E
	Axisymmetric solid two-phase			TAX6P , QAX8P
	Fourier ring			TAX3F , TAX6F , QAX4F , QAX8F
3D Continuum	Solid continuum		TH4 , PN6 , HX8 , HX8M	TH10 , PN12 , PN15 , HX16 , HX20 , TH10S , PN6L , PN12L , HX8L , HX16L , TH4E , PN6E , HX8E
	Solid continuum crack tip			TH10K , PN15K , HX20K
	Solid continuum two phase			TH10P , PN12P , PN15P , HX16P , HX20P
Plates	Isoflex plates		TF3 , QF4 , QSC4	
	Mindlin plates		TTF6 , QTF8	
Shells	Axisymmetric shells		BXS3	

Element Group	Element Subgroup	Element Name and Software Product Version Availability		
		LT	Standard	Plus
	Flat thin shells		TS3 , QSI4	TSR6 ,
	Semiloof shells			TSL6 , QSL8
	Thick shells		TTS3 , QTS4	TTS6 , QTS8
Membranes	Axisymmetric membranes		BXM2 , BXM3	
	Space membranes		TSM3 , SMI4	
Joints	2D joints		JNT3 , JPH3 , JF3 , JAX3 , JXS3	
	3D joints		JNT4 , JL43 , JSH4 , JL46	JSL4
Field	Thermal bars		BFD2 , BFD3 , BFX2 , BFX3 , BFS2 , BFS3	
	Thermal links		LFD2 , LFX2 , LFS2	
	Plane field		TFD3 , TFD6 , QFD4 , QFD8	
	Axisymmetric field		TXF3 , TXF6 , QXF4 , QXF8	
	Solid field		TF4 , TF10 , PF6 , PF12 , PF15 , HF8	HF16 , HF20 , PF6C , PF12C , HF8C , HF16C , TF10S
Interface	2D Interface			IPN4 , IPN6 , IAX4 , IAX6
	3D Interface			IS6 , IS8 , IS12 , IS16
Mass	Point Mass			PM2 , PM3
	Line Mass			LM2 , LM3 , LMS3 , LMS4
	Surface Mass			TM3 , TM6 , QM4 , QM8
Rigid Surface	2D Rigid			R2D2
	3D Rigid			R3D3 , R3D4

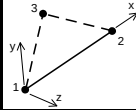
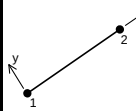
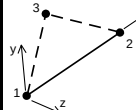
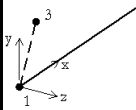
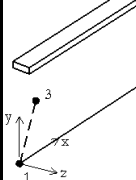
Element Index

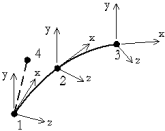
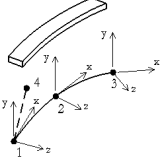
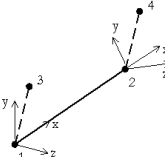
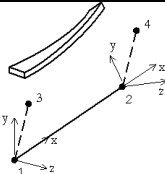
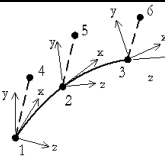
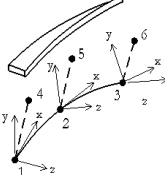
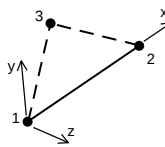
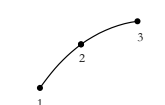
The following element index tables provide a diagrammatic index for each element with a description of the element, the nodal freedoms, and the software product version in which it is available.

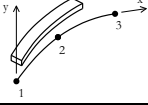
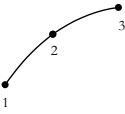
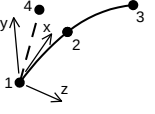
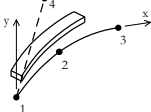
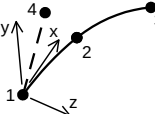
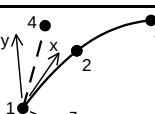
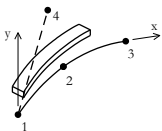
The tables are listed in the following order:

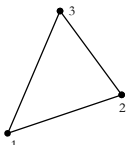
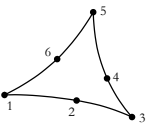
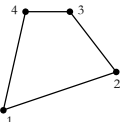
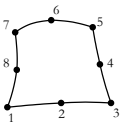
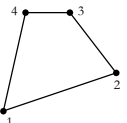
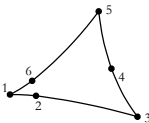
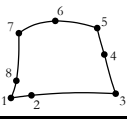
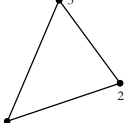
- ☐ [Bar elements](#)
- ☐ [Beam elements](#)
- ☐ [2D Continuum elements](#)
- ☐ [3D Continuum elements](#)
- ☐ [Plate elements](#)
- ☐ [Shell elements](#)
- ☐ [Membrane elements](#)
- ☐ [Joint elements](#)
- ☐ [Thermal / Field elements](#)
- ☐ [Interface elements](#)
- ☐ [Non-Structural Mass elements](#)
- ☐ [Rigid elements](#)
- ☐

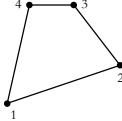
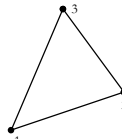
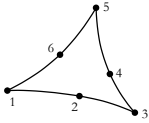
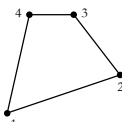
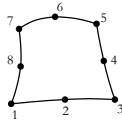
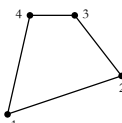
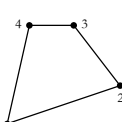
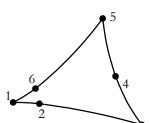
Bar Elements				
Name	Geometry	Title	Freedoms	Product Version
<u>BAR2</u>		BAR element in 2D	U, V	LT
<u>BAR3</u>		BAR element in 2D	U, V	Standard
<u>BRS2</u>		BAR element in 3D	U, V, W	LT
<u>BRS3</u>		BAR element in 3D	U, V, W	Standard

Beam Elements				
Name	Geometry	Title	Freedoms	Product Version
<u>BEAM</u>		ENGINEERING thick beam element in 2D	U, V, qz	LT
<u>BMS3</u>		ENGINEERING thick beam element in 3D	U, V, W, qx, qy, qz	LT
<u>GRIL</u>		ENGINEERING grillage thick beam element in 2D	W, qx, qy	LT
<u>BTS3</u>		THICK beam element in 3D (co-rotational)	U, V, W, qx, qy, qz	Standard
<u>BMI21</u>		THICK linear thick beam element in 3D	U, V, W, qx, qy, qz	LT
<u>BMX21</u>		THICK linear thick beam element in 3D with quadrilateral cross-section	U, V, W, qx, qy, qz	Standard

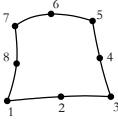
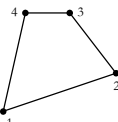
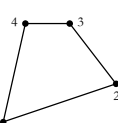
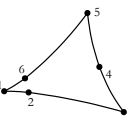
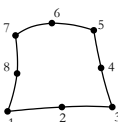
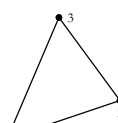
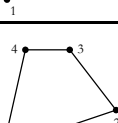
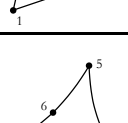
Beam Elements				
Name	Geometry	Title	Freedoms	Product Version
<u>BMI31</u>		THICK quadratic thick beam element in 3D	U, V, W, qx, qy, qz	Plus
<u>BMX31</u>		THICK quadratic thick beam element in 3D with quadrilateral cross-section	U, V, W, qx, qy, qz	Plus
<u>BMI22</u>		THICK twisted linear thick beam element in 3D	U, V, W, qx, qy, qz	Plus
<u>BMX32</u>		THICK twisted linear thick beam element in 3D with quadrilateral cross-section	U, V, W, qx, qy, qz	Plus
<u>BMI33</u>		THICK twisted quadratic thick beam element in 3D with quadrilateral cross-section	U, V, W, qx, qy, qz	Plus
<u>BMX33</u>		THICK beam element in 3D	U, V, W, qx, qy, qz	Plus
<u>BTS3</u>		THICK beam element in 3D (co-rotational)	U, V, W, qx, qy, qz	Standard
<u>BM3</u>		KIRCHHOFF thin beam element in 2D	end nodes: U, V, qz mid-node: dU	Standard

Beam Elements				
Name	Geometry	Title	Freedoms	Product Version
<u>BMX3</u>		KIRCHHOFF thin beam element in 2D with quadrilateral cross-section	end nodes: U, V, qz mid-node: dU	Standard
<u>BS3</u>		KIRCHHOFF thin beam element in 3D	end nodes: U, V, W, qx, qy, qz mid-node: dU, dqx	Plus
<u>BS4</u>		KIRCHHOFF thin beam element in 3D	end nodes: U, V, W, qx, qy, qz mid-node: dU, dqx	Plus
<u>BSX4</u>		KIRCHHOFF thin beam element in 3D with quadrilateral cross-section	end nodes: U, V, W, qx, qy, qz mid-node: dU, dqx	Plus
<u>BSL3</u>		SEMILOOF thin beam element in 3D for use with TSL6	end nodes: U, V, W, qx, qy, qz mid-node: U, V, W, q1, q2	Plus
<u>BSL4</u>		SEMILOOF thin beam element in 3D for use with QSL8	end nodes: U, V, W, qx, qy, qz mid-node: U, V, W, q1, q2	Plus
<u>BXL4</u>		SEMILOOF thin beam element in 3D with quadrilateral cross-section	end nodes: U, V, W, qx, qy, qz mid-node: U, V, W, q1, q2	Plus

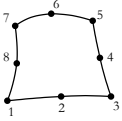
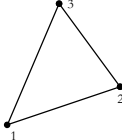
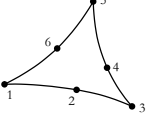
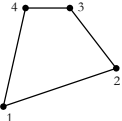
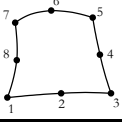
2D Continuum Elements				
Name	Geometry	Title	Freedoms	Product Version
<u>TPM3</u>		PLANE STRESS continuum element in 2D	U, V	Standard
<u>TPM6</u>		PLANE STRESS continuum element in 2D	U, V	Standard
<u>QPM4</u>		PLANE STRESS continuum element in 2D	U, V	Standard
<u>QPM8</u>		PLANE STRESS continuum element in 2D	U, V	Standard
<u>QPM4M</u>		PLANE STRESS continuum element in 2D with enhanced strains	U, V	Standard
<u>TPK6</u>		PLANE STRESS continuum crack tip element in 2D	U, V	Standard
<u>QPK8</u>		PLANE STRESS continuum crack tip element in 2D	U, V	Standard
<u>TPM3E</u>		PLANE STRESS explicit dynamics element in 2D	U, V	Plus

2D Continuum Elements				
Name	Geometry	Title	Freedoms	Product Version
<u>QPM4E</u>		PLANE STRESS explicit dynamics element in 2D	U, V	Plus
<u>TPN3</u>		PLANE STRAIN continuum element in 2D	U, V	Standard
<u>TPN6</u>		PLANE STRAIN continuum element in 2D	U, V	Standard
<u>QPN4</u>		PLANE STRAIN continuum element in 2D	U, V	Standard
<u>QPN8</u>		PLANE STRAIN continuum element in 2D	U, V	Standard
<u>QPN4M</u>		PLANE STRAIN continuum element in 2D with enhanced strains	U, V	Standard
<u>QPN4L</u>		PLANE STRAIN continuum element in 2D for large strains	U, V	Standard
<u>TNK6</u>		PLANE STRAIN continuum crack tip element in 2D	U, V	Standard

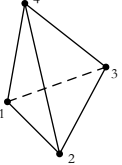
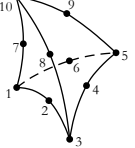
2D Continuum Elements				
Name	Geometry	Title	Freedoms	Product Version
<u>QNK8</u>		PLANE STRAIN continuum crack tip element in 2D	U, V	Standard
<u>TPN3E</u>		PLANE STRAIN explicit dynamics element in 2D	U, V	Plus
<u>QPN4E</u>		PLANE STRAIN explicit dynamics element in 2D	U, V	Plus
<u>TPN6P</u>		PLANE STRAIN continuum two phase element in 2D	U, V P: corner nodes U, V: Midside nodes	Standard
<u>QPN8P</u>		PLANE STRAIN continuum two phase element in 2D	U, V P: corner nodes U, V: Midside nodes	Standard
<u>TAX3</u>		AXISYMMETRIC solid continuum element in 2D	U, V	Standard
<u>TAX6</u>		AXISYMMETRIC solid continuum element in 2D	U, V	Standard
<u>QAX4</u>		AXISYMMETRIC solid continuum element in 2D	U, V	Standard

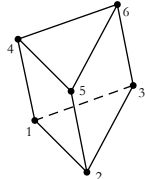
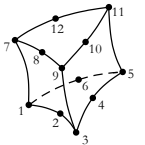
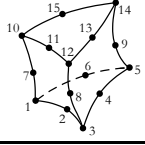
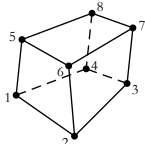
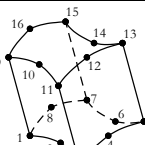
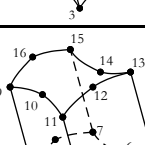
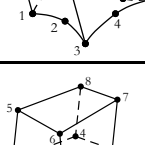
2D Continuum Elements				
Name	Geometry	Title	Freedom	Product Version
<u>QAX8</u>		AXISYMMETRIC solid continuum element in 2D	U, V	Standard
<u>QAX4M</u>		AXISYMMETRIC solid continuum element in 2D with enhanced strains	U, V	Standard
<u>QAX4L</u>		AXISYMMETRIC solid continuum element in 2D for large strains	U, V	Standard
<u>TXK6</u>		AXISYMMETRIC solid continuum crack tip element in 2D	U, V	Standard
<u>QXK8</u>		AXISYMMETRIC solid continuum crack tip element in 2D	U, V	Standard
<u>TAX3E</u>		AXISYMMETRIC solid explicit dynamics element in 2D	U, V	Plus
<u>QAX4E</u>		AXISYMMETRIC solid explicit dynamics element in 2D	U, V	Plus
<u>TAX6P</u>		AXISYMMETRIC solid two phase continuum element in 2D	U, V P: corner nodes U, V: Midside nodes	Plus

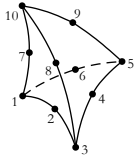
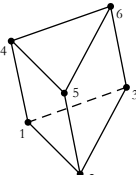
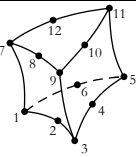
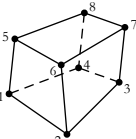
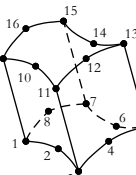
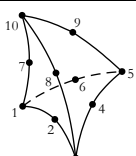
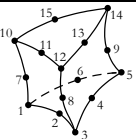
2D Continuum Elements

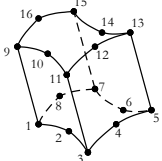
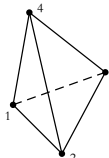
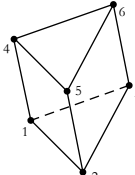
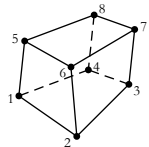
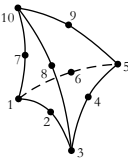
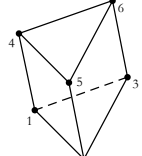
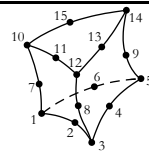
Name	Geometry	Title	Freedoms	Product Version
<u>QAX8P</u>		AXISYMMETRIC solid two phase continuum element in 2D	U, V P: corner nodes U, V: Midside nodes	Plus
<u>TAX3F</u>		AXISYMMETRIC Fourier ring element in 2D	U, V, W	Plus
<u>TAX6F</u>		AXISYMMETRIC Fourier ring element in 2D	U, V, W	Plus
<u>QAX4F</u>		AXISYMMETRIC Fourier ring element in 2D	U, V, W	Plus
<u>QAX8F</u>		AXISYMMETRIC Fourier ring element in 2D	U, V, W	Plus

3D Continuum Elements

Name	Geometry	Title	Freedoms	Product Version
<u>TH4</u>		SOLID CONTINUUM element in 3D	U, V, W	Standard
<u>TH10</u>		SOLID CONTINUUM element in 3D	U, V, W	Plus

3D Continuum Elements				
Name	Geometry	Title	Freedoms	Product Version
<u>PN6</u>		SOLID CONTINUUM element in 3D	U, V, W	Standard
<u>PN12</u>		SOLID CONTINUUM element in 3D	U, V, W	Plus
<u>PN15</u>		SOLID CONTINUUM element in 3D	U, V, W	Plus
<u>HX8</u>		SOLID CONTINUUM element in 3D	U, V, W	Standard
<u>HX16</u>		SOLID CONTINUUM element in 3D	U, V, W	Plus
<u>HX20</u>		SOLID CONTINUUM element in 3D	U, V, W	Plus
<u>HX8M</u>		SOLID CONTINUUM element in 3D with enhanced strains	U, V, W	Standard

3D Continuum Elements				
Name	Geometry	Title	Freedoms	Product Version
<u>TH10S</u>		SOLID CONTINUUM composite element in 3D	U, V, W	Plus
<u>PN6L</u>		SOLID CONTINUUM composite element in 3D	U, V, W	Plus
<u>PN12L</u>		SOLID CONTINUUM composite element in 3D	U, V, W	Plus
<u>HX8L</u>		SOLID CONTINUUM composite element in 3D	U, V, W	Plus
<u>HX16L</u>		SOLID CONTINUUM composite element in 3D	U, V, W	Plus
<u>TH10K</u>		SOLID CONTINUUM crack tip element in 3D	U, V, W	Plus
<u>PN15K</u>		SOLID CONTINUUM crack tip element in 3D	U, V, W	Plus

3D Continuum Elements				
Name	Geometry	Title	Freedoms	Product Version
<u>HX20K</u>		SOLID CONTINUUM crack tip element in 3D	U, V, W	Plus
<u>TH4E</u>		SOLID CONTINUUM explicit dynamics element in 3D	U, V, W	Plus
<u>PN6E</u>		SOLID CONTINUUM explicit dynamics element in 3D	U, V, W	Plus
<u>HX8E</u>		SOLID CONTINUUM explicit dynamics element in 3D	U, V, W	Plus
<u>TH10P</u>		SOLID CONTINUUM two phase element in 3D	U, V, W	Plus
<u>PN12P</u>		SOLID CONTINUUM two phase element in 3D	U, V, W	Plus
<u>PN15P</u>		SOLID CONTINUUM two phase element in 3D	U, V, W	Plus

3D Continuum Elements

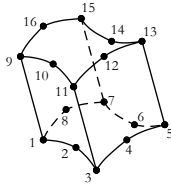
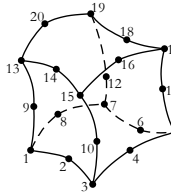
Name	Geometry	Title	Freedoms	Product Version
<u>HX16P</u>		SOLID CONTINUUM two phase element in 3D	U, V, W	Plus
<u>HX20P</u>		SOLID CONTINUUM two phase element in 3D	U, V, W	Plus

Plate Elements

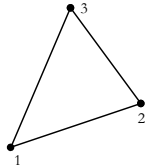
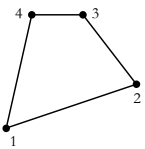
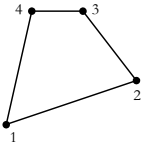
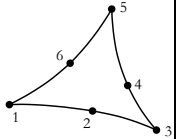
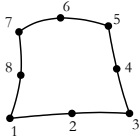
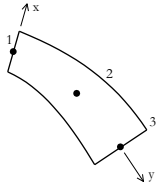
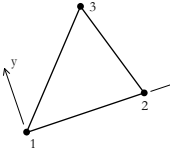
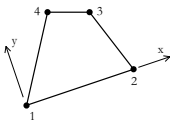
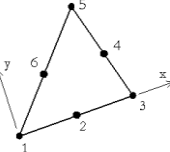
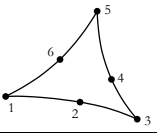
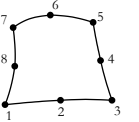
Name	Geometry	Title	Freedoms	Product Version
<u>TF3</u>		ISOFLEX thin plate flexure element in 2D	W, qx, qy	Standard
<u>QF4</u>		ISOFLEX thin plate flexure element in 2D	W, qx, qy	Standard
<u>QSC4</u>		ISOFLEX thick plate flexure element in 2D	W, qx, qy	Standard
<u>TTF6</u>		MINDLIN thick plate flexure element in 2D	W, qx, qy	Standard

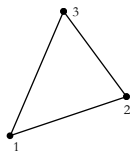
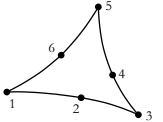
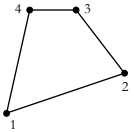
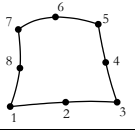
Plate Elements

Name	Geometry	Title	Freedoms	Product Version
<u>QTF8</u>		MINDLIN thick plate flexure element in 2D	W, qx, qy	Standard

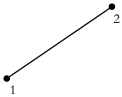
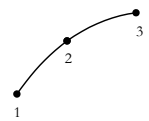
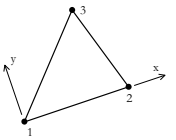
Shell Elements

Name	Geometry	Title	Freedoms	Product Version
<u>BXS3</u>		AXISYMMETRIC thin shell element in 2D	end nodes: U, V, qz mid-node: dU	Standard
<u>TS3</u>		FLAT thin shell element in 3D	U, V, W, qx, qy, qz	Standard
<u>QSI4</u>		FLAT thin shell element in 3D	U, V, W, qx, qy, qz	Standard
<u>TSR6</u>		FLAT thin nonlinear shell element in 3D	corner nodes: U, V, W mid-side nodes: q1	Plus
<u>TSL6</u>		SEMILOOF curved thin shell element in 3D	corner nodes: U, V, W mid-side nodes: U, V, W, q1, q2	Plus
<u>QSL8</u>		SEMILOOF curved thin shell element in 3D	corner nodes: U, V, W mid-side nodes: U, V, W, q1, q2	Plus

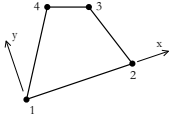
Shell Elements

Name	Geometry	Title	Freedoms	Product Version
<u>TTS3</u>		THICK SHELL flat element in 3D	U, V, W, qa, qbor U, V, W, qx, qy, qz	Standard
<u>TTS6</u>		THICK SHELL curved element in 3D	U, V, W, qa, qbor U, V, W, qx, qy, qz	Plus
<u>QTS4</u>		THICK SHELL flat element in 3D	U, V, W, qa, qbor U, V, W, qx, qy, qz	Standard
<u>QTS8</u>		THICK SHELL curved element in 3D	U, V, W, qa, qbor U, V, W, qx, qy, qz	Plus

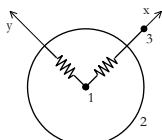
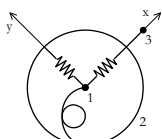
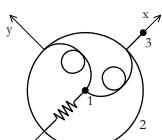
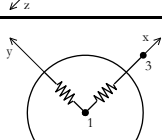
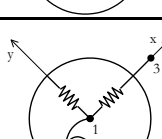
Membrane Elements

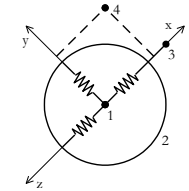
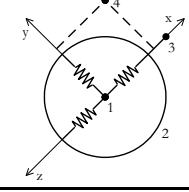
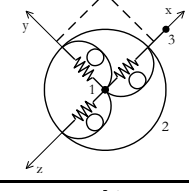
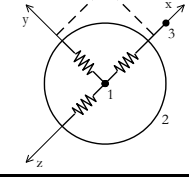
Name	Geometry	Title	Freedoms	Product Version
<u>BXM2</u>		AXISYMMETRIC membrane element in 2D	U, V	Standard
<u>BXM3</u>		AXISYMMETRIC membrane element in 2D	U, V	Standard
<u>TSM3</u>		SPACE membrane element in 3D	U, V, W	Standard

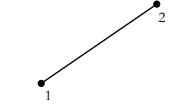
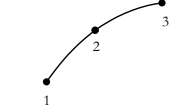
Membrane Elements

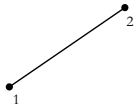
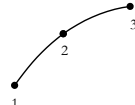
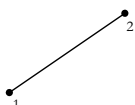
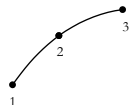
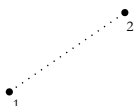
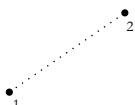
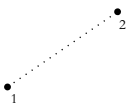
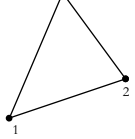
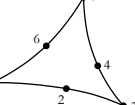
Name	Geometry	Title	Freedoms	Product Version
<u>SMI4</u>		SPACE membrane element in 3D	U, V, W	Standard

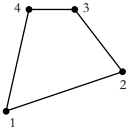
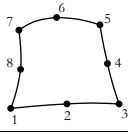
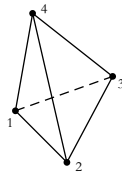
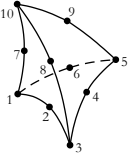
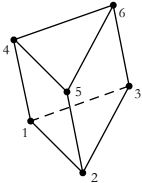
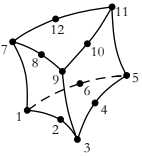
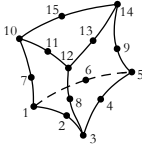
Joint Elements

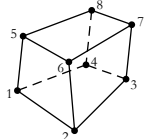
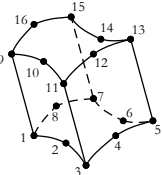
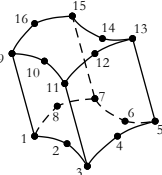
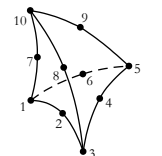
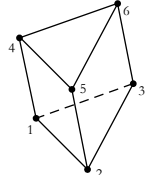
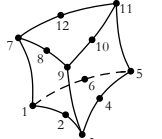
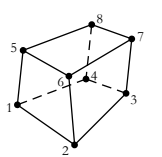
Name	Geometry	Title	Freedoms	Product Version
<u>JNT3</u>		JOINT ELEMENT in 2D for bars, plane stress and plane strain	U, V	Standard
<u>JPH3</u>		JOINT ELEMENT in 2D for engineering and Kirchhoff beams	U, V, qz	Standard
<u>JE3</u>		JOINT ELEMENT in 2D for grillage beams and plates	W, qx, qy	Standard
<u>JAX3</u>		JOINT ELEMENT in 2D for axisymmetric solids	U, V	Standard
<u>JXS3</u>		JOINT ELEMENT in 2D for axisymmetric shells	U, V, qz	Standard

Joint Elements				
Name	Geometry	Title	Freedoms	Product Version
<u>JNT4</u>		JOINT ELEMENT in 3D for bars, solids and space membranes	U, V, W	Standard
<u>JL43</u>		JOINT ELEMENT in 3D for corner nodes of semiloof elements	U, V, W	Standard
<u>JSH4</u> <u>JL46</u>		JOINT ELEMENT in 3D for engineering and Kirchhoff beams and the end/corner nodes of semiloof elements	U, V, W, qx, qy, qz	Standard
<u>JSL4</u>		JOINT ELEMENT in 3D for mid-side nodes of semiloof elements	U, V, W, q1, q2	Plus

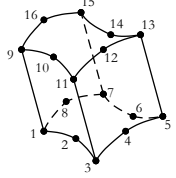
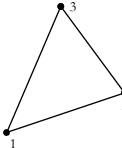
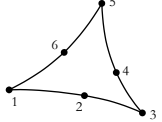
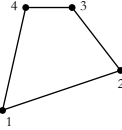
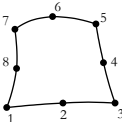
Thermal / Field Elements				
Name	Geometry	Title	Freedoms	Product Version
<u>BFD2</u>		THERMAL BAR element in 2D	F	Standard
<u>BFD3</u>		THERMAL BAR element in 2D	F	Standard

Thermal / Field Elements				
Name	Geometry	Title	Freedom	Product Version
<u>BFX2</u>		Axisymmetric THERMAL MEMBRANE element in 2D	F	Standard
<u>BFX3</u>		Axisymmetric THERMAL MEMBRANE element in 2D	F	Standard
<u>BFS2</u>		THERMAL BAR element in 3D	F	Standard
<u>BFS3</u>		THERMAL BAR element in 3D	F	Standard
<u>LFD2</u>		THERMAL LINK element in 2D	F	Standard
<u>LFX2</u>		Axisymmetric THERMAL LINK element in 2D	F	Standard
<u>LFS2</u>		THERMAL LINK element in 3D	F	Standard
<u>TFD3</u>		PLANE FIELD element in 2D	F	Standard
<u>TFD6</u>		PLANE FIELD element in 2D	F	Standard

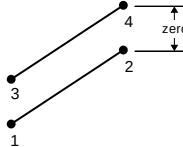
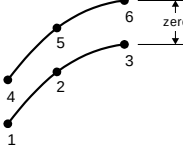
Thermal / Field Elements				
Name	Geometry	Title	Freedoms	Product Version
<u>QFD4</u>		PLANE FIELD element in 2D	F	Standard
<u>QFD8</u>		PLANE FIELD element in 2D	F	Standard
<u>TF4</u>		SOLID FIELD element in 3D	F	Standard
<u>TF10</u>		SOLID FIELD element in 3D	F	Plus
<u>PF6</u>		SOLID FIELD element in 3D	F	Standard
<u>PF12</u>		SOLID FIELD element in 3D	F	Plus
<u>PF15</u>		SOLID FIELD element in 3D	F	Plus

Thermal / Field Elements				
Name	Geometry	Title	Freedoms	Product Version
<u>HF8</u>		SOLID FIELD element in 3D	F	Standard
<u>HF16</u>		SOLID FIELD element in 3D	F	Plus
<u>HF20</u>		SOLID FIELD element in 3D	F	Plus
<u>TF10S</u>		SOLID FIELD composite element in 3D	F	Plus
<u>PF6C</u>		SOLID FIELD composite element in 3D	F	Plus
<u>PF12C</u>		SOLID FIELD composite element in 3D	F	Plus
<u>HF8C</u>		SOLID FIELD composite element in 3D	F	Plus

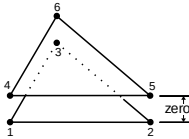
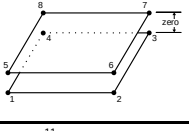
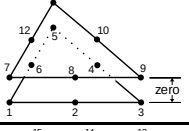
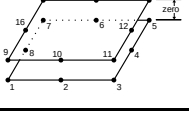
Thermal / Field Elements

Name	Geometry	Title	Freedoms	Product Version
<u>HF16C</u>		SOLID FIELD composite element in 3D	F	Plus
<u>TXF3</u>		AXISYMMETRIC FIELD element in 2D	F	Standard
<u>TXF6</u>		AXISYMMETRIC FIELD element in 2D	F	Standard
<u>QXF4</u>		AXISYMMETRIC FIELD element in 2D	F	Standard
<u>QXF8</u>		AXISYMMETRIC FIELD element in 2D	F	Standard

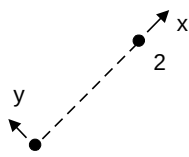
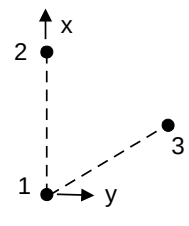
Interface Elements

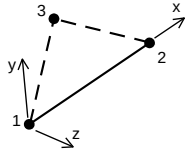
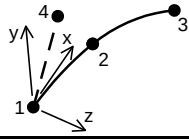
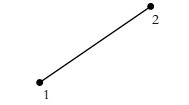
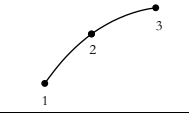
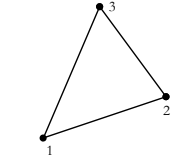
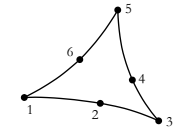
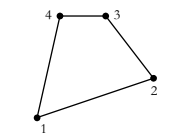
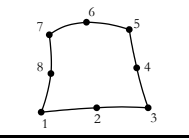
Name	Geometry	Title	Freedoms	Product Version
<u>IPN4</u> <u>IAx4</u>		INTERFACE ELEMENT in 2D for modelling delamination and axisymmetric crack propagation	U, V	Plus
<u>IPN6</u> <u>IAx6</u>		INTERFACE ELEMENT in 2D for modelling delamination and axisymmetric crack propagation	U, V	Plus

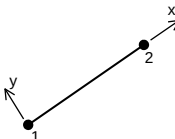
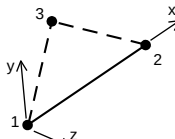
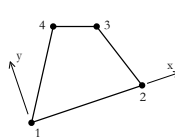
Interface Elements

Name	Geometry	Title	Freedoms	Product Version
<u>IS6</u>		INTERFACE ELEMENT in 3D for modelling delamination and crack propagation	U, V, W	Plus
<u>IS8</u>		INTERFACE ELEMENT in 3D for modelling delamination and crack propagation	U, V, W	Plus
<u>IS12</u>		INTERFACE ELEMENT in 3D for modelling delamination and crack propagation	U, V, W	Plus
<u>IS16</u>		INTERFACE ELEMENT in 3D for modelling delamination and crack propagation	U, V, W	Plus

Non-Structural Mass Elements

Name	Geometry	Title	Freedoms	Product Version
<u>PM2</u>		NON-STRUCTURAL MASS ELEMENT in 2D to model mass at a point	U, V	Plus
<u>PM3</u>		NON-STRUCTURAL MASS ELEMENT in 3D to model mass at a point	U, V, W	Plus

Non-Structural Mass Elements				
Name	Geometry	Title	Freedoms	Product Version
<u>LMS3</u>		NON-STRUCTURAL MASS ELEMENT in 3D to model mass along an edge	U, V, W	Plus
<u>LMS4</u>		NON-STRUCTURAL MASS ELEMENT in 3D to model mass along an edge	U, V, W	Plus
<u>LM2</u>		NON-STRUCTURAL MASS ELEMENT in 2D to model mass along an edge	U, V	Plus
<u>LM3</u>		NON-STRUCTURAL MASS ELEMENT in 2D to model mass along an edge	U, V	Plus
<u>TM3</u>		NON-STRUCTURAL MASS ELEMENT in 3D to model mass on a surface.	U,V,W	Plus
<u>TM6</u>		NON-STRUCTURAL MASS ELEMENT in 3D to model mass on a surface.	U,V,W	Plus
<u>QM4</u>		NON-STRUCTURAL MASS ELEMENT in 3D to model mass on a surface.	U,V,W	Plus
<u>QM8</u>		NON-STRUCTURAL MASS ELEMENT in 3D to model mass on a surface.	U,V,W	Plus

Rigid Elements				
Name	Geometry	Title	Freedoms	Product Version
<u>R2D2</u>		RIGID SURFACE ELEMENT in 2D for modelling non-deformable surfaces in a contact analysis	U, V	Plus
<u>R3D3</u>		RIGID SURFACE ELEMENT in 3D for modelling non-deformable surfaces in a contact analysis	U, V, W	Plus
<u>R3D4</u>		RIGID SURFACE ELEMENT in 3D for modelling non-deformable surfaces in a contact analysis	U, V, W	Plus

Element Summary Tables

The following element summary tables list element facilities arranged by LUSAS element group:

- ☐ [Bar and Beam elements](#)
- ☐ [2D Continuum elements](#)
- ☐ [3D Continuum elements](#)
- ☐ [Plate, Shell and Membrane elements](#)
- ☐ [Joint elements](#)
- ☐ [Thermal / Field elements](#)
- ☐ [Interface, Non-Structural Mass and Rigid elements](#)

Bar and Beam Element Summary

Bar and Beam Element Summary		Bars		Beams															
		BAR2, BAR3	BRS2, BRS3	BEAM	BMS3	GRIL	BTS3	BMI21	BMX21	BMI22	BMI31	BMX22,1	BM3	BMX3	BS3, BS4	BSX4	BSL3, BSL4	BXL4	
Product version	LT, Standard (S) or Plus (+)	LT	LT	LT	LT	LT	S	LT	S	+	+	S	S	+	+	+	+		
Nodal Freedoms (mid-side)	U, V	✓																	
	U, V, W		✓																
	U, V, qz			✓															
	U, V, qz (dU)												✓	✓					
	W, qx, qy						✓												
	U, V, W, qx, qy, qz (dU, dqx)														✓	✓			
	U, V, W, qx, qy, qz (U, V, W,q1, q2)																✓	✓	
Material Properties	U, V, W, qx, qy																		
	U, V, W, qx, qy, qz				✓		✓	✓	✓	✓	✓								
	Linear (Isotropic)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		
	Linear (Orthotropic)																		
	Linear (Anisotropic)																		
	Linear (Rigidities)						✓	✓		✓		✓		✓	✓	✓			
	Matrix																		
	Joint																		
	Concrete																		
	Stress Resultant						✓	✓		✓		✓		✓		✓			
	Tresca	✓	✓						✓		✓		✓		✓		✓		
	Drucker-Prager	✓	✓						✓		✓		✓		✓		✓		
	Mohr-Coulomb	✓	✓						✓		✓		✓		✓		✓		
	Optimised Implicit Von Mises	✓	✓						✓		✓		✓		✓		✓		
	Volumetric Crushing/Foam																		
	Stress Potential(Von Mises, Modified Von Mises)	✓	✓						✓		✓		✓		✓		✓		
	Creep (General)	✓	✓				✓		✓		✓		✓		✓		✓		
	Creep (CEB-FIP)						✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	
	Creep (Chinese)						✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	
	Creep (Eurocode)						✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	
	Damage (Simo, Oliver)	✓	✓						✓		✓		✓		✓		✓		
	Viscoelastic	✓																	
	Shrinkage	✓						✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	

Bar and Beam Element Summary

Bar and Beam Element Summary		Bars		Beams													
		BAR2, BAR3	BRS2, BRS3	BEAM	BMS3	GRIL	BTS3	BMI21	BMX21	BMI22	BMI31	BMX22	BM3	BMX3	BS3, BS4	BSX4	BSL3, BSL4
Product version	LT, Standard (S) or Plus (+)	LT	LT	LT	LT	LT	S	LT	S	+	+	S	S	+	+	+	+
Loading Types	(CEB-FIP_90, Eurocode_2, General, User)																
	Rubber																
	Generic Polymer																
	Multi-linear	✓	✓														
	Composite																
	Field																
	Prescribed Value (PDSP,TPDSP)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Concentrated Loads (CL)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Element Load (ELDS)			✓	✓	✓	✓	✓	✓	✓	✓	✓	✓				✓
	Distributed Load (UDL)			✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Nonlinear Geometry	Distributed Load (FLD)																
	Body Force (CBF)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Body Force (BFP,BFPE)	✓	✓					✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Velocity (VELO)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Acceleration (ACCE)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Initial Stress/Strain (SSI,SSIE)	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Initial Stress/Strain (SSIG)	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Residual Stress (SSR,SSRE)						✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Residual Stress (SSRG)	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Temperature (TEMP,TMPE)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Field Loads																
	Temperature Dependent Loads																
	Total Lagrangian	✓	✓					✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Updated Lagrangian												✓	✓	✓	✓	
	Eulerian																
	Integration Schemes	Co-rotational						✓									
Explicitly Integrated				✓	✓	✓											
Numerically Integrated		✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Consistent Mass (default)		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Mass Modelling	Lumped Mass	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

2D Continuum Element Summary

2D Continuum Element Summary		2D Continuum																
		TPM3/6, QPM4/8	QPM4M	TPK6, QPK8	TPM3E, QPM4E	TPN3/6, QPN4/8	QPN4M	QPN4L	TNK6, QNK8	TPN3E, QPN4E	TPN6P, QPN8P	TAX3/6, QAX4/8	QAX4M	QAX4L	TXK6, QXK8	TAX3E, QAX4E	TAX6P, QAX8P	TAX3F/6F, QAX4F/8F
Product Version	LT, Standard (S) or Plus (+)	S	S	S	+	S	S	S	S	+	+	S	S	S	S	+	+	+
Nodal Freedoms (corner) Material Properties	U, V	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓	✓	✓	✓	✓		
	U, V, W																	✓
	U, V, (P)										✓						✓	
	Linear (Isotropic)	✓	✓	✓	✓	✓	✓		✓	✓	✓	✓	✓		✓	✓	✓	✓
	Linear (Orthotropic)	✓	✓	✓	✓	✓	✓		✓	✓	✓	✓	✓		✓	✓	✓	✓
	Linear (Anisotropic)	✓	✓	✓		✓	✓*		✓*		✓*	✓	✓*		✓*		✓*	
	Linear (Rigidities)	✓	✓	✓		✓	✓*		✓*		✓*						✓*	
	Matrix																	
	Joint																	
	Concrete (Multi-crack)	✓	✓	✓		✓	✓		✓		✓	✓	✓		✓		✓	
	Stress Resultant																	
	Tresca	✓	✓	✓	✓	✓	✓		✓	✓	✓	✓	✓		✓	✓	✓	
	Optimised Implicit Von Mises	✓	✓	✓	✓	✓	✓		✓	✓	✓	✓	✓		✓	✓	✓	
	Mohr-Coulomb	✓	✓	✓	✓	✓	✓		✓	✓	✓	✓	✓		✓	✓	✓	
	Modified Mohr-Coulomb					✓	✓		✓	✓	✓	✓	✓		✓	✓	✓	
	Drucker-Prager	✓	✓	✓	✓	✓	✓		✓	✓	✓	✓	✓		✓	✓	✓	
	Modified Cam-clay					✓	✓		✓		✓	✓	✓		✓		✓	
	Volumetric Crushing/Foam					✓	✓		✓	✓	✓	✓	✓		✓	✓	✓	
	Stress Potential (Von Mises, Modified Von Mises)	✓	✓	✓	✓	✓	✓		✓	✓	✓	✓	✓		✓	✓	✓	
	Interface (2D)	✓																
	Creep (General)	✓	✓	✓			✓		✓	✓	✓		✓		✓	✓	✓	
	Creep (CEB-FIP)	✓	✓	✓		✓	✓		✓			✓	✓		✓	✓		
	Creep (Chinese)	✓	✓	✓		✓	✓		✓			✓	✓		✓	✓		

2D Continuum Element Summary

2D Continuum Element Summary		2D Continuum																	
		TPM3/6, QPM4/8	QPM4M	TPK6, QPK8		TPM3E, QPM4E	TPN3/6, QPN4/8	QPN4M	QPN4L	TNK6, QNK8	TPN3E, QPN4E	TPN6P, QPN8P	TAX3/6, QAX4/8	QAX4M	QAX4L	TXK6, QXK8	TAX3E, QAX4E	TAX6P, QAX8P	TAX3F/6F, QAX4F/8F
Loading Types	Creep (Eurocode)	✓	✓	✓		✓	✓		✓			✓	✓		✓	✓			
	Damage (Simo, Oliver)	✓	✓	✓		✓	✓		✓	✓	✓	✓	✓		✓	✓	✓		
	Viscoelastic					✓	✓				✓	✓	✓	✓		✓	✓	✓	
	Shrinkage (CEB-FIP, Eurocode. General, User)	✓		✓		✓	✓		✓			✓	✓	✓	✓	✓		✓	
	Rubber (Ogden, Mooney-Rivlen, Neo-Hookean, Hencky)		✓				✓	✓						✓					
	Generic Polymer		✓	✓		✓	✓		✓			✓	✓	✓		✓		✓	
	Composite																		
	Field																		
	Prescribed Value (PDSP,TPDSP)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Concentrated Loads (CL)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Element Load																		
	Distributed Load (UDL)																		
	Distributed Load (FLD)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Body Force (CBF,BFP,BFPE)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Velocity (VELO)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Acceleration (ACCE)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Initial Stress/Strain (SSI,SSIE)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Initial Stress/Strain (SSIG)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓	✓	✓
	Residual Stress (SSR)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	
	Residual Stress (SSRE,SSRG)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	
Temperature	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	

2D Continuum Element Summary

2D Continuum Element Summary		2D Continuum																	
		TPM3/6, QPM4/8	QPM4M	TPK6, QPK8	TPM3E, QPM4E	TPN3/6, QPN4/8	OPN4M	QPN4L	TNK6, QNK8	TPN3E, QPN4E	TPN6P, QPN8P	TAX3/6, QAX4/8	QAX4M	QAX4L	TXK6, QXK8	TAX3E, QAX4E	TAX6P, QAX8P	TAX3F/6F, QAX4F/8F	
Nonlinear Geometry	(TEMP,TMPE)																		
	Field Load																		
	Temp Dependent Load																		
	Total Lagrangian	✓	✓	✓		✓	✓		✓		✓	✓	✓		✓		✓		
	Updated Lagrangian	✓	✓	✓		✓	✓		✓		✓	✓	✓		✓		✓		
	Eulerian	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		
	Co-rotational	✓	✓	✓		✓	✓		✓		✓								
Integration Schemes	Explicitly Integrated																		
	Numerically Integrated	✓	✓	✓	✓	✓	✓	✓	✓		✓	✓	✓	✓	✓	✓	✓	✓	
Mass Modelling	Consistent Mass (default)	✓	✓	✓		✓	✓	✓	✓		✓	✓	✓	✓	✓		✓	✓	
	Lumped Mass	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	

3D Continuum Element Summary

3D Continuum Element Summary		3D Continuum												
		TH4	TH10	PN6	PN12/15	HX8	HX16/20	HX8M	TH10K, PN15K, HX20K	TH10S	PN6L, PN12L	HX8L, HX16L	TH4E, PN6E, HX8E	TH10P, PN12P, PN15P, HX16P, HX20P
Product Version	LT, Standard (S) or Plus (+)	S	+	S	+	S	+	S	+	+	+	+	+	+
Nodal Freedoms (corner) Material Properties	U, V													
	U, V, W	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	
	U, V, W (P)													✓
	Linear (Isotropic)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Linear (Orthotropic)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Linear (Anisotropic)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓
	Linear (Rigidities)													
	Matrix													
	Joint													
	Concrete (Multi-crack)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓
	Stress Resultant													
	Tresca	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Optimised Implicit Von Mises	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
		Mohr-Coulomb	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Modified Mohr-Coulomb		✓	✓	✓	✓	✓	✓	✓	✓					✓
Drucker-Prager		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Modified Cam-clay		✓	✓	✓	✓	✓	✓	✓	✓					✓
Volumetric Crushing/Foam		✓	✓	✓	✓	✓	✓		✓	✓	✓	✓	✓	✓
Stress Potential(Von Mises, Modified Von Mises Hill, Hoffman)		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Creep (General)		✓	✓	✓	✓	✓	✓		✓				✓	✓
Creep (CEB-FIP)		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		
Creep (Chinese)		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		
Creep (Eurocode)		✓	✓	✓		✓	✓		✓			✓	✓	

3D Continuum Element Summary

3D Continuum Element Summary		3D Continuum												
		TH4	TH10	PN6	PN12/15	HX8	HX16/20	HX8M	TH10K, PN15K, HX20K	TH10S	PN6L, PN12L	HX8L, HX16L	TH4E, PN6E, HX8E	TH10P, PN12P, PN15P, HX16P, HX20P
Loading Types	Damage	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Viscoelastic	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓
	Shrinkage (CEB-FIP, Eurocode, General, User)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓
	Elasto-plastic interface	✓	✓	✓	✓	✓	✓		✓					✓
	Rubber (Ogden, Mooney-Rivlin, Neo- Hookean, Hencky							✓						
	Generic Polymer	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓
	Resin Cure Model								.	✓	✓	✓		
	Composite (Composite Solid)								.	✓	✓	✓		
	Composite (Composite Shell)													
	Field													
	Prescribed Value (PDSP,TPDSP)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Concentrated Loads (CL)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Element Loads													
	Distributed Load (UDL)													
	Distributed Load (FLD)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Body Force (CBF,BFP,BFPE)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Velocity (VELO)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Acceleration (ACCE)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Initial Stress/Strain (SSI,SSIE)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Initial Stress/Strain (SSIG)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓
	Residual Stress	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

3D Continuum Element Summary

3D Continuum Element Summary		3D Continuum												
		TH4	TH10	PN6	PN12/15	HX8	HX16/20	HX8M	TH10K, PN15K, HX20K	TH10S	PN6L, PN12L	HX8L, HX16L	TH4E, PN6E, HX8E	TH10P, PN12P, PN15P, HX16P, HX20P
Nonlinear Geometry	(SSR,SSRE)													
	Residual Stress (SSRG)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Temperature (TEMP,TMPE)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Field Load													
	Temp Dependent Load													
	Total Lagrangian	✓	✓	✓	✓	✓	✓	✓	✓					✓
	Updated Lagrangian	✓	✓	✓	✓	✓	✓	✓	✓					✓
	Eulerian	✓	✓	✓	✓	✓	✓	✓	✓				✓	✓
	Co-rotational	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓
Integration Schemes	Explicitly Integrated													
	Numerically Integrated	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Mass	Consistent Mass (default)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓			✓
Modelling	Lumped Mass	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

Plate, Shell and Membrane Element Summary

Plate, Shell and Membrane Element Summary		Plates			Shells							Mem		
		TF3, QF4	QSC4	TTF6, QTF8	BXS3	TS3, QSI4	TSR6	TSL6, QSL8	TTS3	TTS6	QTS4	QTS8	BXM2/3	TSM3, SMI4
Product Version	LT, Standard (S) or Plus (+)	S	S	S	S	S	+	+	S	+	S	+	S	S
Nodal Freedoms (mid-side)	U, V												✓	
	U, V, W													✓
	W, qx, qy		✓	✓										
	W, qx, qy (dq)	✓												
	U, V, W, qx, qy													
	U, V, qz (dU)				✓									
	U, V, W, qx, qy, qz					✓								
	U, V, W (U, V, W, q1, q2)							✓						
	U, V, W (q1,)						✓							
Material Properties	U, V, W, qa, qb (U, V, W, qx, qy, qz)								✓	✓	✓	✓		
	Linear (Isotropic)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Linear (Orthotropic)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓
	Linear (Anisotropic)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓
	Linear (Rigidities)	✓	✓	✓		✓	✓	✓						✓
	Matrix													
	Joint													
	Concrete (Multi-crack)						✓	✓	✓	✓	✓	✓		
	Stress Resultant				✓		✓	✓						
	Tresca				✓		✓	✓	✓	✓	✓	✓	✓	
	Optimised Implicit Von Mises				✓		✓	✓	✓	✓	✓	✓	✓	
	Mohr-Coulomb				✓		✓	✓	✓	✓	✓	✓	✓	
	Drucker-Prager				✓		✓	✓	✓	✓	✓	✓	✓	
	Volumetric Crushing/Foam													
	Stress Potential (Von-Mises, Modified Von Mises)				✓		✓	✓	✓	✓	✓	✓	✓	
	Stress Potential(Hill, Hoffman)				✓		✓	✓	✓	✓	✓	✓		
Creep (General)				✓		✓	✓	✓	✓	✓	✓	✓		

Plate, Shell and Membrane Element Summary

Plate, Shell and Membrane Element Summary		Plates			Shells								Mem	
		TF3, QF4	QSC4	TTF6, QTF8	BXS3	TS3, QSI4	TSR6	TSL6, QSL8	TIS3	TIS6	QTS4	QTS8	BXM2I3	TSM3, SMI4
Loading Types	Creep (CEB_FIP_90)				✓			✓	✓	✓	✓	✓		
	Creep (Chinese)				✓			✓	✓	✓	✓	✓		
	Creep (Eurocode)				✓			✓	✓	✓	✓	✓		
	Damage				✓		✓	✓	✓	✓	✓	✓	✓	
	Viscoelastic													
	Shrinkage (CEB-FIP_90, Eurocode_2, General, User)				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Rubber (Ogden, Mooney-Rivlin, Neo-Hookean, Hencky)												✓	
	Generic Polymer													
	Composite (Composite Shell)							✓	✓	✓	✓	✓		
	Field													
	Prescribed Value (PDSP,TPDSP)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Concentrated Loads (CL)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Element Load (ELDS)				✓									
	Distributed Load (UDL)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓
	Distributed Load (FLD)				✓								✓	
	Body Force (CBF,BFP,BFPE)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Velocity (VELO)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Acceleration (ACCE)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Initial Stress/Strain (SSI,SSIE)	✓	✓	✓	✓	✓	✓						✓	✓
	Initial Stress/Strain (SSIG)				✓		✓	✓	✓	✓	✓	✓	✓	✓
	Residual Stress (SSR,SSRE)						✓							
	Residual Stress (SSRG)				✓		✓	✓	✓	✓	✓	✓	✓	
	Temperature (TEMP,TMPE)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Field Load													

Plate, Shell and Membrane Element Summary

Plate, Shell and Membrane Element Summary		Plates			Shells							Mem		
		TF3, QF4	QSC4	TTF6, QTF8	BXS3	TS3, QSI4	TSR6	TSL6, QSL8	TTS3	TTS6	QTS4	QTS8	BXM2/3	TSM3, SMI4
Nonlinear Geometry	Temp Dependent Loads													
	Total Lagrangian				✓		✓	✓	✓	✓	✓	✓		
	Updated Lagrangian				✓		✓	✓						
	Eulerian													
Integration Schemes	Co-rotational					✓								
	Explicitly Integrated													
	Numerically Integrated	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	
Mass	Consistent Mass (default)	✓	✓	✓	✓		✓	✓	✓	✓	✓	✓		
	Lumped Mass	✓	✓	✓	✓	✓		✓	✓	✓	✓	✓	✓	

Joint Element Summary

		Joints							
		JNT3	JPH3	JF3	JAX3	JXS3	JNT4, JL43	JSH4, JL46	JSL4
Product Version	LT, Standard (S) or Plus (+)	S	S	S	S	S	S	S	+
Nodal Freedoms	U, V	✓			✓				
	U, V, W						✓		
	U, V, qz		✓			✓			
	W, qx, qy			✓					
	U, V, W, qx, qy								
	U, V, W, qx, qy, qz							✓	
	U, V, W, q1, q2								✓
Material Properties	Linear								
	Matrix (Stiffness, Mass, Damping)*	✓	✓	✓	✓	✓	✓	✓	✓
	Joint (Stiffness, General)	✓	✓	✓	✓	✓	✓	✓	✓
	Joint (Dynamic, General)	✓	✓	✓	✓	✓	✓	✓	✓
	Joint (Elasto-Plastic)	✓	✓	✓	✓	✓	✓	✓	✓
	Joint (Nonlinear Contact)	✓	✓	✓	✓	✓	✓	✓	✓
	Joint (Nonlinear Friction)	✓	✓		✓	✓	✓	✓	✓
	Viscous damping	✓	✓	✓	✓	✓	✓	✓	✓
	Lead-Rubber	✓	✓	✓	✓	✓	✓	✓	✓
	Friction Pendulum	✓	✓	✓	✓	✓	✓	✓	✓
	Multilinear elastic	✓	✓	✓	✓	✓	✓	✓	✓
	Axial force dependent multilinear elastic	✓	✓	✓	✓	✓	✓	✓	✓
	Concrete								
	Elasto-Plastic								
	Creep								
	Damage								
	Viscoelastic								
	Shrinkage								
	Volumetric Crushing/Foam								
	Rubber								
	Composite								
	Field								
Loading	Prescribed value	✓	✓	✓	✓	✓	✓	✓	✓

Joint Element Summary		Joints							
		JNT3	JPH3	JF3	JAX3	JXS3	JNT4, JL43	JSH4, JL46	JSL4
Types	(PDSP,TPDSP)								
	Concentrated Load (CL)	✓	✓	✓	✓	✓	✓	✓	✓
	Element Load								
	Distributed Load								
	Body Force(CBF)	✓	✓	✓	✓	✓	✓	✓	✓
	Body Force (BFP,BFPE)								
	Velocities (VELO)	✓	✓	✓	✓	✓	✓	✓	✓
	Acceleration (ACCE)	✓	✓	✓	✓	✓	✓	✓	✓
	Initial Stress/Strain (SSI,SSIE)	✓	✓	✓	✓	✓	✓	✓	✓
	Initial Stress/Strain (SSIG)								
	Residual Stress								
	Temperature (TEMP,TMPE)	✓	✓	✓	✓	✓	✓	✓	✓
	Field Load								
	Temp Dependent Load								
Nonlinear Geometry	Total Lagrangian								
	Updated Lagrangian								
	Eulerian								
	Co-rotational								
	Explicitly Integrated								
Integration Schemes	Numerically Integrated	✓	✓	✓	✓	✓	✓	✓	✓
	Consistent Mass (default)								
	Lumped Mass	✓	✓	✓	✓	✓	✓	✓	✓

* Supported in LUSAS Solver but not supported in LUSAS Modeller for all joints listed.

Thermal / Field Element Summary

Thermal / Field Element Summary		Field																		
		BFD2/3	BFX2/3	BFS2/3	LFD2	LFX2	LFS2	TFD3/6, QFD4/8	TFX3/6, QFX4/8	TF4	TF10	PF6	PF12/15	HF8	HF16/20	TF10S	PF6C, HF8C	PF12C, HF16C	TXF3, QXF4	TXF6, OXF8
Product Version	LT, Standard (S) or Plus (+)	S	S	S	S	S	S	S	S	S	S	S	S	S	+	+	+	+	S	S
Freedoms Material Properties	F	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Linear																			
	Matrix																			
	Joint																			
	Concrete																			
	Elasto-Plastic																			
	Creep																			
	Damage																			
	Viscoelastic																			
	Shrinkage																			
	Rubber																			
	Generic Polymer																			
	Composite															✓	✓	✓		
	Field (Isotropic)		✓	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Field (Isotropic Concrete)		✓	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Field (Orthotropic)								✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Field (Orthotropic Concrete)								✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Field (Linear Conv/Rad)					✓	✓	✓													
Field (Arbitrary Conv/Rad)					✓	✓	✓													
Loading Types	Prescribed Value (PDSP,TPDSP)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Concentrated Loads (CL)	✓	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Element Load																			
	Distributed Load (UDL)															✓	✓	✓	✓	✓
	Distributed Load (FLD)	✓	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Body Force (CBF, BFP, BFPE)	✓	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Velocity																			
	Acceleration																			

Thermal / Field Element Summary

Thermal / Field Element Summary		Field																		
		BFD2/3	BFX2/3	BFS2/3	LFD2	LFX2	LFS2	TFD3/6, QFD4/8	TFX3/6, QFX4/8	TF4	TF10	PF6	PF12/15	HF8	HF16/20	TF10S	PF6C, HF8C	PF12C, HF16C	TXF3, QXF4	TXF6, QXF8
Nonlinear Geometry	Initial Stress/Strain																			
	Residual Stress																			
	Temperature (TEMP, TMPE)	✓	✓	✓																
	Field Load (ENVT)	✓	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Temp Dep Load (TDET/RIHG)	✓	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Total Lagrangian																			
	Updated Lagrangian																			
	Eulerian																			
	Co-rotational																			
Integration Schemes Specific Heat	Explicitly Integrated																			
	Numerically Integrated	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Consistent (default)	✓	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Lumped	✓	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

Interface, Non-Structural Mass and Rigid Element Summary

Interface, Non-Structural Mass and Rigid Element Summary		Interface				Mass				Rigid		
		IPN4, IAX4	IPN6, IAX6	IS6, IS8	IS12, IS16	PM2	PM3	LMS3, LMS4	LM2, LM3	TM3/6, QM4/8	R2D2	R3D3, R4D3
Product Version	LT, Standard (S) or Plus (+)	+	+	+	+	+	+	+	+	+	+	+
Nodal Freedoms	U, V	✓	✓			✓			✓		✓	
	U, V, W			✓	✓		✓	✓		✓		✓
	U, V, qz											
	W, qx, qy											
	U, V, W, qx, qy											
Material Properties	U, V, W, qx, qy, qz											
	U, V, W, q1, q2											
	Linear										✓	✓
	Matrix											
	Joint											
	Mass					✓	✓	✓	✓	✓		
	Concrete											
	Elasto-Plastic											
	Creep											
	Damage											
	Shrinkage											
	Interface	✓	✓	✓	✓							
	Rubber											
	Generic Polymer											
	Stress Potential											
Loading Types	Composite											
	Field											
	Prescribed value (PDSP,TPDSP)	✓	✓	✓	✓						✓	✓
	Concentrated Loads (CL)	✓	✓	✓	✓							
	Element Load											
	Distributed Load											
	Body Force (CBF)					✓	✓	✓	✓	✓		
Body Force (BFP,BFPE)												

Interface, Non-Structural Mass and Rigid Element Summary

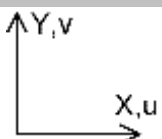
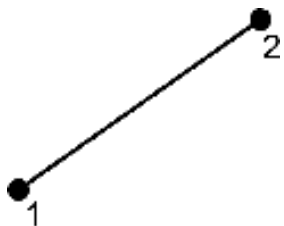
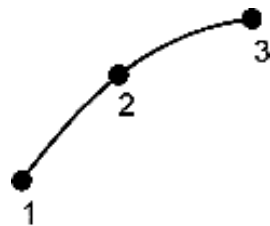
		Interface				Mass					Rigid	
		IPN4, IAX4	IPN6, IAX6	IS6, IS8	IS12, IS16	PM2	PM3	LMS3, LMS4	LM2, LM3	TM3/6, QM4/8	R2D2	R3D3, R4D3
Nonlinear Geometry	Velocity (VELO)	✓	✓	✓	✓						✓	✓
	Acceleration (ACCE)	✓	✓	✓	✓						✓	✓
	Initial Stress/Strain (SSI, SSIE)											
	Initial Stress/Strain (SSIG)											
	Residual Stress											
	Temperature (TEMP, TMPE)	✓	✓	✓	✓							
	Field Load											
	Temp Dependent Load											
	Total Lagrangian										✓*	✓*
	Updated Lagrangian										✓*	✓*
	Eulerian										✓*	✓*
	Co-rotational	✓		✓							✓*	✓*
Integration Schemes	Explicitly Integrated											
	Numerically Integrated	✓	✓	✓	✓	✓	✓	✓	✓	✓		
Mass Modelling	Consistent Mass (default)					✓	✓	✓	✓	✓		
	Lumped Mass					✓	✓	✓	✓	✓		

* Dependent upon the other surface (deformable surface) that the element is in contact with.

Chapter 1 : Bar Elements.

2D Structural Bar Elements

General

Element Name	BAR2	BAR3
		
Element Group	Bars	
Element Subgroup	Structural Bars	
Element Description	Straight and curved isoparametric bar elements in 2D which can accommodate varying cross sectional area.	
Number Of Nodes	2 or 3.	
Freedom	U, V at each node	
Node Coordinates	X, Y at each node	

Geometric Properties

$A_1 \dots A_n$ Cross sectional area at each node.

Material Properties

Linear	Isotropic	MATERIAL PROPERTIES (Elastic: Isotropic)
Matrix	Not applicable	
Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic	Stress resultant	Not applicable
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)

	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	Not applicable
	Stress Potential	STRESS POTENTIAL VON_MISES (Isotropic: von Mises, Modified von Mises)
Creep		CREEP PROPERTIES (Creep)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic		VISCO ELASTIC PROPERTIES
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable	
Multi-linear		MATERIAL PROPERTIES NONLINEAR 104
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V at each node.
Concentrated Loads	CL	Concentrated loads. Px, Py at each node.
Element Loads	Not applicable.	
Distributed Loads	Not applicable.	
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Ω_x , Ω_y , Ω_z , α_z
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, 0, Xcbf, Ycbf
Velocities	VELO	Velocities. Vx, Vy at nodes.
Accelerations	ACCE	Acceleration Ax, Ay at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. (1) Resultants (linear material models): Fx , ϵ_x (2) Components (nonlinear material models): 0, 0, σ_x , ϵ_x
	SSIG	Initial stresses/strains at Gauss points. (1) Resultants (linear material models): Fx , ϵ_x

		(2) Components (nonlinear model): 0, 0, σ_x , ϵ_x
Residual Stresses	SSR, SSRE SSRG	Not applicable. Residual stresses at Gauss points. Components (nonlinear material models): 0, 0, σ_x
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, T ₀ , 0, 0, 0 in local directions.
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Force (default): F _x Strain: ϵ_x
Modeller	See Results Tables (Appendix K)

Local Axes

- ☐ [Standard line element](#)

Sign Convention

- ☐ [Standard bar element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	For large displacements and small strains
Updated Lagrangian	Not applicable.
Eulerian	Not applicable.
Co-rotational	Not applicable.

Integration Schemes

Stiffness	Default. Fine (see <i>Options</i>).	1-point (BAR2), 2-point (BAR3). 2-point (BAR2).
Mass	Default. Fine (see	2-point (BAR2), 3-point (BAR3). As default.

Options).

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes fine integration rule for element.
- 55** Outputs strains as well as stresses
- 87** Total Lagrangian geometric nonlinearity.
- 105** Lumped mass matrix.

Notes on Use

1. The bar formulation is based on the standard [isoparametric](#) approach. The variation of axial force is constant for BAR2, and linear for BAR3.
2. Since the 3-noded element has no bending stiffness mechanisms may occur when used as 'stand alone' elements if the central node is not constrained in some way.
3. When the BAR2 element is used with either varying cross-sectional area or temperature dependent material properties, the 2-point Gauss rule should be utilised. This provides an improved representation of the variation of the material properties along the length of the element.

Restrictions

- ☐ [Ensure mid-side node centrality](#)
- ☐ Avoid excessive element curvature

Recommendations on Use

- The 2-node elements are the most effective bar elements for modelling 'stand-alone-elements' such as members of trusses or bars connecting two discrete structures.
- They can be used to model cables in cable-stayed structures.
- Both the 2-noded and 3-noded elements are suitable for modelling reinforcement with continuum elements e.g. BAR3 may be used with QPM8 for analysis of reinforced concrete structures, or for modelling rock bolts surrounding an excavation

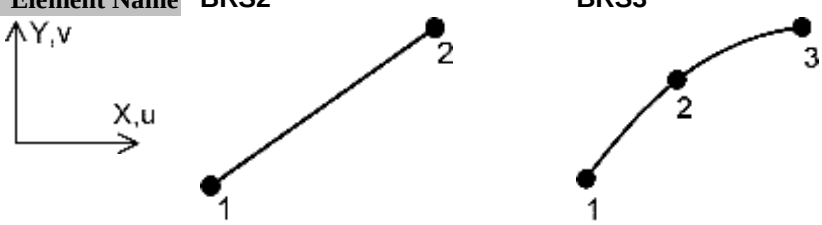
Theory

For additional information see the *LUSAS Theory Manual*

3D Structural Bar Elements

General

Element Name	BRS2	BRS3



Element Group	Bars
Element Subgroup	Structural Bars
Element Description	Straight and curved isoparametric bar elements in 3D which can accommodate varying cross-sectional area.
Number Of Nodes	2 or 3.
Freedom	U, V, W at each node
Node Coordinates	X, Y, Z at each node.

Geometric Properties

$A_1 \dots A_n$ Cross sectional area at each node.

Material Properties

Linear	Isotropic	MATERIAL PROPERTIES (Elastic: Isotropic)
Matrix	Not applicable	
Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic	Stress resultant	Not applicable
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65

		(Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	Not applicable
	Stress Potential	STRESS POTENTIAL VON_MISES (Isotropic: von Mises, Modified von Mises)
Creep		CREEP PROPERTIES (Creep)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic		VISCO ELASTIC PROPERTIES
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Multi-linear		MATERIAL PROPERTIES NONLINEAR 104
Rubber	Not applicable	
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W at each node.
Concentrated Loads	CL	Concentrated loads. Px, Py, Pz at each node.
Element Loads	Not applicable	
Distributed Loads	Not applicable	
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf, Ω_x , Ω_y , Ω_z , α_x , α_y , α_z
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, 0, Xcbf, Ycbf, Zcbf
Velocities	VELO	Velocities. Vx, Vy, Vz at nodes.
Accelerations	ACCE	Acceleration Ax, Ay, Az at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. (1) Resultants (linear material models): Fx , ϵ_x (2) Components (nonlinear material models): 0, 0, σ_x , ϵ_x
	SSIG	Initial stresses/strains at Gauss points. (1) Resultants (linear material models): Fx , ϵ_x (2) Components (nonlinear model): 0, 0, σ_x ,

		ϵ_x
Residual Stresses	SSR, SSRE SSRG	Not applicable Residual stresses at Gauss points. Components (nonlinear material models): 0, 0, σ_x
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0 in local directions.
Field Loads	Not applicable	
Temp Dependent Loads	Not applicable	

LUSAS Output

Solver	Force (default): Fx Strain: ϵ_x
Modeller	See Results Tables (Appendix K)

Local Axes

- ☐ [Standard line element](#)

Sign Convention

- ☐ [Standard bar element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	For large displacements and small strains
Updated Lagrangian	Not applicable.
Eulerian	Not applicable.
Co-rotational	Not applicable.

Integration Schemes

Stiffness	Default.	1-point (BRS2), 2-point (BRS3).
	Fine (see <i>Options</i>).	2-point (BRS2).
Mass	Default.	2-point (BRS2), 3-point (BRS3).
	Fine (see <i>Options</i>).	As default.

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes fine integration rule for element.
- 55** Outputs strains as well as stresses
- 87** Total Lagrangian geometric nonlinearity.
- 105** Lumped mass matrix.

Notes on Use

1. The bar formulation is based on the standard
2. Since the 3-noded element has no bending stiffness, mechanisms may occur, when used as 'stand alone' elements, if the central node is not constrained in some way.
3. When the BRS2 element is used with either varying cross-sectional area or temperature dependent material properties, the 2-point Gauss rule should be utilised. This provides an improved representation of the variation of the material properties along the length of the element.

Restrictions

- ☐ [Ensure mid-side node centrality](#)
- ☐ Avoid excessive element curvature

Recommendations on Use

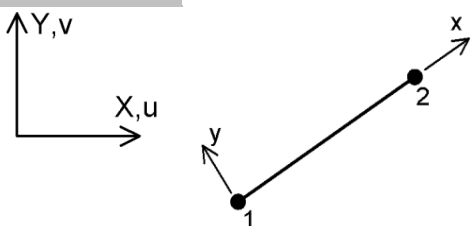
- The 2-node elements are the most effective bar elements for modelling 'stand-alone-elements' such as members of trusses or bars connecting two discrete structures.
- They can be used to model cables in cable-stayed structures.
- Both the 2-noded and 3-noded elements are suitable for modelling reinforcement with continuum elements e.g. BRS3 may be used with HX20 for analysis of reinforced concrete structures, or for modelling rock bolts surrounding an excavation.

Chapter 2 : Beam Elements.

2D Engineering Thick Beam Element

General

Element Name	BEAM



Element Group	Beams
Element Subgroup	Engineering Beams
Element Description	A straight beam element for which shearing deformations are included. The geometric properties are constant along the length.
Number Of Nodes	2 with moment release end conditions.
Freedom	U, V, θ_z : at each node.
Node Coordinates	The element node numbers should be followed by: R restrained (default), F free defined in the order θ_z at node 1 and then θ_z at node 2 related to local element axes.

Geometric Properties

A, Izz, Asy, for element

ey

A Cross sectional area

Izz 2nd moment of area about local z-axis (see [Definition](#))

Asy Effective [shear area](#) on local yz plane in local y directions

ey [Eccentricity](#) from beam xz-plane to nodal line (+ve in +ve local y-direction)

Material Properties

Linear Isotropic: MATERIAL PROPERTIES (Elastic: Isotropic)

Matrix Not applicable

Joint Not applicable

Concrete Not applicable

Elasto-Plastic Not applicable

Creep	Not applicable
Damage	Not applicable
Viscoelastic	Not applicable
Shrinkage	Not applicable
Rubber	Not applicable
Generic Polymer	Not applicable
Composite	Not applicable

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, θ_z : at nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py, Mz: at nodes (global).
Element Loads	ELDS	Element loads on nodal line (load type number LTYPE *10 defines the corresponding element load type on beam axis, see Notes) LTYPE, S1, Px, Py, Mz LTYPE=11: point loads and moments in local directions. LTYPE=12: point loads and moments in global directions. LTYPE, 0, Wx, Wy, 0 LTYPE=21: uniformly distributed loads in local directions. LTYPE=22: uniformly distributed loads in global directions. LTYPE=23: uniformly distributed projected loads in global directions LTYPE, S1, Wx1, Wy1, 0, S2, Wx2, Wy2, 0 LTYPE=31: distributed loads in local directions. LTYPE=32: distributed loads in global directions. LTYPE=33: distributed projected loads in global directions LTYPE, S1, Wx, Wy, 0 LTYPE=41: trapezoidal loads in local directions. LTYPE=42: trapezoidal loads in global directions. LTYPE=43: trapezoidal projected loads in global directions
Distributed Loads	UDL	Uniformly distributed loads. Wx, Wy: forces/unit length for element in local directions.

	FLD	Not applicable.
Body Forces	CBF	Constant body forces for element. X_{cbf} , Y_{cbf} , Ω_x , Ω_y , Ω_z , α_z
	BFP, BFPE	Not applicable.
Velocities	VELO	Velocities. V_x , V_y : at nodes.
Accelerations	ACCE	Acceleration. A_x , A_y : at nodes.
Initial Stress/Strains		Not applicable.
Residual Stresses		Not applicable.
Temperatures	TEMP, TMPE	Temperatures at nodes/for elements. T , 0, dT/dy , 0, T_0 , 0, dT_0/dy , 0
Field Loads		Not applicable.
Temp Dependent Loads		Not applicable.

LUSAS Output

Solver	Force (default): F_x , F_y , M_z : in local directions. Element output is with respect to the beam centre line.
Modeller	See Results Tables (Appendix K) .

Local Axes

- ☐ [Standard line element](#)

Sign Convention

- ☐ [2D engineering beam element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

- [Explicitly integrated.](#)

Mass Modelling

- Consistent mass (default).
- Lumped mass.

Options

- 105** Lumped mass matrix
- 380** Output stress/strain resultants with respect to the beam centroidal axes.
- 381** Input beam geometric properties and apply CBF, UDL, SSI and TEMP loads along centroidal beam axes
- 405** Specify geometric properties along beam centroidal axes
- 406** Specify CBF, UDL, SSI, SSR and TEMP loads along beam centroidal axes (see Notes)
- 418** Output stress resultants relative to beam centroidal axes for eccentric elements (see Notes)

Note: OPTION 380 is on by default for this element. For output with respect to the nodal line specify OPTION -380.

Notes on Use

1. The beam formulation is based on the standard engineering beam approach. The force variations along the beam are constant axial force, linear shear forces and linear moments; however, a quadratic variation in bending moment will be obtained if a distributed load is applied to an element. Note that a quadratic variation in bending moment cannot be achieved in an eigenvalue analysis
2. The displacement variations along the beam are linear axial, linear rotation, and cubic transverse displacement.
3. Strains are not available with this element.
4. Though this element cannot model nonlinear behaviour, it can be mixed with other elements in a nonlinear analysis.
5. Internal forces and moments are output at intervals of 1/10th of the element length by specifying the Gauss point option from the Output button on the **File > LUSAS datafile** dialog.
6. If a step by step dynamic analysis is carried out (with damping) using BEAM elements with distributed loading, the force diagrams would not be correct until all inertia forces have been damped out and the beam reaches a state of static equilibrium. For this reason, the “free body force diagrams” pertaining to applied loading, are not included in the analysis. This means that different force diagrams will now be obtained for BEAM elements if static and dynamic analyses are directly compared. Therefore, in a dynamic analysis, BEAM

elements will perform in the same manner as other beam elements in LUSAS where free body force diagrams are never included.

7. If OPTION 416 is false, this element always outputs stress/strain resultants with respect to beam centroidal axes; if OPTION 416 is true, output will be controlled by OPTION 418.
8. OPTION 406 will replace OPTION 381. If OPTION 381 set TRUE in the dat file OPTION 406 will automatically be set TRUE within Solver.
9. ELDS loading on the beam axis can be specified in two ways: multiply the load type number LTYPE by 10, or set OPTION 381 true, e.g. trapezoidal projected load on the beam axis can be specified by ITYPE = 430 or ITYPE = 43 with OPTION 381.

Restrictions

The element does not model material or geometric nonlinear effects.

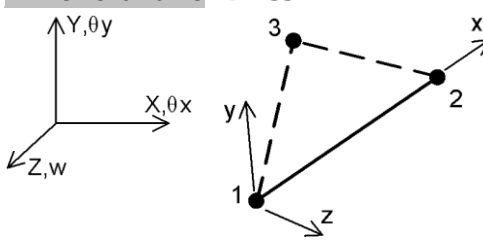
Recommendations on Use

- The element can be used to model one and two dimensional beam and frame structures using linear, eigen, and dynamic analysis procedures.
- The element has constant material properties along its length. For analyses utilising temperature dependent material properties, the temperature used is the average of the nodal values.
- A moment release option permits modelling of internal hinges. For further information see [Number of Nodes](#) at the top of this page.
- The consistent mass is formulated directly from engineering beam theory and incorporates rotational inertia. The lumped mass matrices are evaluated using the procedure defined in the LUSAS Theory Manual

3D Engineering Thick Beam Element

General

Element Name	BMS3



Element Group	Beams
Element Subgroup	Engineering Beams
Element Description	A straight beam element in 3D for which shear deformations are included. The geometric properties are constant along the length.
Number Of Nodes	3 with end release conditions. The third node is used to define the local xy-plane.
Freedom	U, V, W, θ_x , θ_y , θ_z : at end nodes.
End Releases	The element node numbers should be followed by: R restrained (default), F free defined in the order θ_y , θ_z at node 1 and then θ_y , θ_z at node 2 related to local element axes
Node Coordinates	X, Y, Z: at each node.

Geometric Properties

A, Iyy, Izz, Jxx, Asz, for element

Asy, ez, ey

A Cross sectional area

Iyy, Izz Moments of inertia about local y, z axes (see [Definition](#))

Jxx [Torsional constant](#)

Asz, Asy Effective shear areas on local yz plane in local z, y directions (see [Shear Areas](#))

ez [Eccentricity](#) from beam xy-plane to nodal line. (+ve in +ve local z-direction)

ey [Eccentricity](#) from beam xz-plane to nodal line. (+ve in +ve local y-direction)

Material Properties

Linear	Isotropic:
Matrix	Not applicable
Joint	Not applicable
Concrete	Not applicable
Elasto-Plastic	Not applicable
Creep	Not applicable
Damage	Not applicable
Viscoelastic	Not applicable
Shrinkage	Not applicable
Rubber	Not applicable
Generic Polymer	Not applicable
Composite	Not applicable

MATERIAL PROPERTIES (Elastic: Isotropic)

Loading

Prescribed Value	PDSP, TPDSP
Concentrated Loads	CL
Element Loads	ELDS

Prescribed variable. U, V, W, θ_x , θ_y , θ_z at nodes. Concentrated loads in global directions. Px, Py, Pz, Mx, My, Mz: at nodes.

[Element loads](#) on nodal line (load type number LTYPE *10 defines the corresponding element load type on beam axis, see Notes)

LTYPE, S1, Px, Py, Pz, Mx, My, Mz

LTYPE=11: point loads and moments in local directions.

LTYPE=12: point loads and moments in global directions.

LTYPE, 0, Wx, Wy, Wz, Mx, 0, 0

LTYPE=21: uniformly distributed loads in local directions.

LTYPE=22: uniformly distributed loads in global directions (Mx=0).

LTYPE=23: uniformly distributed projected loads in global directions (Mx=0).

LTYPE, S1, Wx1, Wy1, Wz1, Mx1, 0, 0,

S2, Wx2, Wy2, Wz2, Mx2, 0, 0

LTYPE=31: distributed loads in local directions.

LTYPE=32: distributed loads in global directions (Mx1=0, Mx2=0).

LTYPE=33: distributed projected loads

		in global directions ($M_{x1}=0$, $M_{x2}=0$).
		LTYPE, S1, W_x , W_y , W_z , M_x , 0, 0
		LTYPE=41: trapezoidal loads in local directions.
		LTYPE=42: trapezoidal loads in global directions ($M_x=0$).
		LTYPE=43: trapezoidal projected loads in global directions ($M_x=0$).
Distributed Loads	UDL	Uniformly distributed loads. W_x , W_y , W_z : forces/unit length for element in local directions.
	FLD	Not applicable.
Body Forces	CBF	Constant body forces for Element. X_{cbf} , Y_{cbf} , Z_{cbf} , Ω_x , Ω_y , Ω_z , α_x , α_y , α_z
	BFP, BFPE	Not applicable.
Velocities	VELO	Velocities. V_x , V_y , V_z : At Nodes.
Accelerations	ACCE	Acceleration A_x , A_y , A_z : At Nodes.
Initial Stress/Strains		Not applicable.
Residual Stresses		Not applicable.
Temperatures	TEMP, TMPE	Temperatures at nodes/for elements. T , 0, dT/dy , dT/dz , T_0 , 0, dT_0/dy , dT_0/dz in local directions.
Field Loads		Not applicable.
Temp Dependent Loads		Not applicable.

LUSAS Output

Solver	Force (default): F_x , F_y , F_z , M_x , M_y , M_z : in local directions. Element output is with respect to the beam centre line.
Modeller	See Results Tables (Appendix K) .

Local Axes

- ☐ [Standard line element](#)

Sign Convention

- ☐ [3D engineering beam element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

[Explicitly integrated.](#)

Mass Modelling

Consistent mass (default).

Lumped mass.

Options

- 105 Lumped mass matrix
- 380 Output stress/strain resultants with respect to the beam centroidal axes.
- 381 Input beam geometric properties and apply CBF, UDL, SSI and TEMP loads along beam centroidal axes
- 405 Specify geometric properties along beam centroidal axes
- 406 Specify CBF, UDL, SSI, SSR and TEMP loads along beam centroidal axes (see Notes)
- 418 Output stress resultants relative to beam centroidal axes for eccentric elements (see Notes)

Note: OPTION 380 is on by default for this element. For output with respect to the nodal line specify OPTION -380.

Notes on Use

1. The element formulation is the standard engineering beam element formulation. The force variations along the beam are constant axial force, constant torsion, linear shear forces and linear moments; however, a quadratic variation in bending moment will be obtained if a distributed load is applied to an element. Note that a quadratic variation in bending moment cannot be achieved in an eigenvalue analysis.
2. Loading that varies along the element length is accounted for in the force diagrams (i.e. for a beam under CBF or internal element loading)
3. The displacement variations along the beam are linear axial, linear rotation, and cubic transverse displacement.
4. Strains are not available with this element.

5. Though this element cannot model nonlinear behaviour, it can be mixed with other elements in a nonlinear analysis.
6. Internal forces and moments are output at intervals of 1/10th of the element length by specifying the Gauss point option from the **Output** button of the LUSAS Datafile dialog.
7. If a step by step dynamic analysis is carried out (with damping) using BMS3 elements with distributed loading, the force diagrams would not be correct until all inertia forces have been damped out and the beam reaches a state of static equilibrium. For this reason, the “free body force diagrams” pertaining to applied loading, are not included in the analysis. This means that different force diagrams will now be obtained for BMS3 elements if static and dynamic analyses are directly compared. Therefore, in a dynamic analysis, BMS3 elements will perform in the same manner as other beam elements in LUSAS where free body force diagrams are never included.
8. If OPTION 416 is false, this element always outputs stress/strain resultants with respect to beam centroidal axes; if OPTION 416 is true, output will be controlled by OPTION 418
9. OPTION 406 will replace OPTION 381. If OPTION 381 set TRUE in the dat file OPTION 406 will automatically be set TRUE within Solver.
10. ELDS loading can be specified on the beam axis in two ways: multiply the load type number LTYPE by 10, or set OPTION 381 true, e.g. trapezoidal projected load on the beam axis can be specified by ITYPE = 430 or ITYPE = 43 with OPTION 381.

Restrictions

The element does not model material or geometric nonlinear effects.

In order to separate the bending behaviour about the beam local y and z axes the standard engineering beam formulation assumes that the beam axes are coincident with the principal axes of the section. This implies that I_{yz} of the section must equal zero.

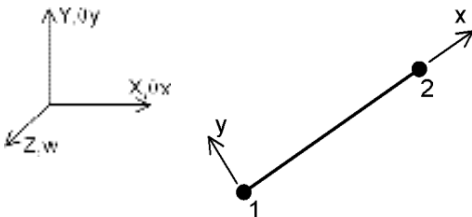
Recommendations on Use

- The element can be used to model 3D frame structures, or act as a stiffener for the [QTS4, TTS3](#) and [QSI4, TS3](#) flat shell elements. Linear, eigen, and dynamic analysis procedures are fully supported with this element.
- The element can be used to model cables in cable-stayed structures.
- The element has constant material properties along its length. For analyses utilising temperature dependent material properties, the temperature used is the average of the nodal values.

- A moment release option permits modelling of internal hinges about the y and z-axes of the local Cartesian system (torsional rotations cannot be released). See [End Releases](#) for further details.
- Consider using BMI21 elements in place of BMS3 elements because they are computationally equivalent and also support material nonlinearity, variations of section properties and twisting.

2D Engineering Grillage Thick Beam Element

General

Element Name	GRIL
	
Element Group	Beams
Element Subgroup	Engineering Beams
Element Description	A straight grillage element for which shear deformations are included. The geometric properties are constant along the length.
Number Of Nodes	2 with moment release end conditions
End Releases	The element node numbers should be followed by: R restrained (default), F free defined in the order θ_y at node 1 and then θ_y at node 2 related to local element axes
Freedom Node Coordinates	w, θ_x, θ_y : at each node. x, y : at each node.

Geometric Properties

$A, I_{yy}, I_{zz}, J_{xx}, A_{sz}$	for element EFW
A	Cross sectional area
I_{yy}, I_{zz}	2nd moments of area about local y, z axes (see Definition and Notes)
J_{xx}	Torsional constant
A_{sz}	Effective shear area on local yz plane in local z directions
EFW	Effective width

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)

Matrix	Not applicable
Joint	Not applicable
Concrete	Not applicable
Elasto-Plastic	Not applicable
Creep	Not applicable
Damage	Not applicable
Viscoelastic	Not applicable
Shrinkage	Not applicable
Rubber	Not applicable
Generic Polymer	Not applicable
Composite	Not applicable.

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. W, θ_x , θ_y : at nodes.
Concentrated Loads	CL	Concentrated loads. Pz, Mx, My: at nodes (global).
Element Loads	ELDS	Element loads LTYPE, S1, Pz, Mx, My LTYPE=11: point loads and moments in local directions. LTYPE=12: point loads and moments in global directions. LTYPE, 0, Wz, Mx, 0 LTYPE=21: uniformly distributed loads in local directions. LTYPE, S1, Wz1, Mx1, 0, S2, Wz2, Mx2, 0 LTYPE=31: distributed loads in local directions. LTYPE, S1, Wz, Mx, 0 LTYPE=41: trapezoidal loads in local directions.
Distributed Loads	UDL	Uniformly distributed loads. Wz: Force/unit length in local directions for element (Local z)

	FLD	and global Z are coincident).
	Not applicable.	
Body Forces	CBF	Constant body forces for element. Zcbf
	BFP, BFPE	Not applicable.
Velocities	VELO	Velocities. Vz: at nodes.
Accelerations	ACCE	Acceleration Az: at nodes.
Initial Stress/Strains	Not applicable.	
Residual Stresses	Not applicable.	
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. 0, 0, 0, dT/dz, 0, 0, 0, dTo/dz: in local directions.
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

Output

Solver	Force (default): Fz, Mx, My: in local directions (see <i>Notes</i>). Element output is with respect to the beam centre line.
Modeller	See Results Tables (Appendix K) .

Local Axes

- ☐ [Standard line element](#)

Sign Convention

- ☐ [2D engineering grillage thick beam element](#). Positive external forces and moments acting on the element nodes are in the direction of the local element axes.

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

[Explicitly integrated.](#)

Mass Modelling

- ☐ Consistent mass (default).

- ☐ Lumped mass.

Options

105 Lumped mass matrix

Notes on Use

1. The element formulation is based on the standard grillage element formulation. The force variations along the element are linear shear force, constant torsion and quadratic bending moment.
2. The displacement variations along the element are linear torsional rotations and cubic transverse flexural displacements.
3. Internal forces and moments are output at intervals of 1/10th of the element length by specifying the Gauss point option from the Output button on the **File** > **LUSAS datafile** dialog.
4. The [second moment of area](#) about local z, (I_{zz}), is only required when assembling the mass matrix.
5. Strains are not available for GRIL elements.
6. Though this element cannot model nonlinear behaviour it can be mixed with other elements in a nonlinear analysis.
7. For restrictions on the use of [Wood-Armer](#) with grillages refer to the LUSAS User Guide and Theory Manual.
8. The element has constant material properties along its length. For analyses utilising temperature dependent material properties, the temperature used is the average of the nodal values.
9. A moment release option permits modelling of internal hinges (torsional rotations cannot be released). See [Number of Nodes](#) section.

Restrictions

The element does not model material or geometric nonlinear effects.

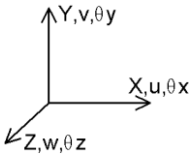
Recommendations on Use

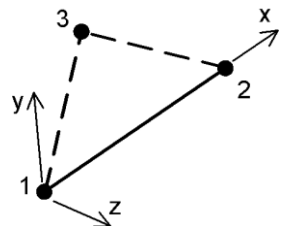
The element can be used to model two dimensional grillage type structures. Linear, eigen, and dynamic analysis procedures can be used with GRIL elements.

3D Thick Beam Element (nonlinear)

General

Element Name	BTS3





Element Group	Beams
Element Subgroup	Thick Beams
Element Description	A straight beam element in 3D for which shear deformations are included. The geometric properties are constant along the length.
Number Of Nodes	3 with end release conditions. The third node is used to define the local xy-plane.
Freedom	U, V, W, θ_x , θ_y , θ_z : at end nodes.
End Releases	The element node numbers should be followed by: R restrained (default), F free defined in the order U, V, W, θ_x , θ_y , θ_z at node 1 and then U, V, W, θ_x , θ_y , θ_z at node 2 related to local element axes, see <i>Notes</i> .)
Node Coordinates	X, Y, Z: at each node.

Geometric Properties

A, Iyy, Izz, Jxx, Asz, Asy, for element.

Iy, Iz, Iyz, ez, ey

A Cross sectional area.

Iyy, Izz Moments of inertia about local y, z axes (see [Definition](#)).

Jxx [Torsional constant](#)

Asz, Asy Effective shear areas on local yz plane in local z, y directions (see [shear areas](#)).

Iy, Iz 1st moment of area about local y, z directions (see [Definition](#)).

Iyz Product moment of inertia about local y, z axes (see [Definition](#)).

ez Eccentricity from beam xy-plane to nodal line. (+ve in the

+ve local z direction). (See Notes)

ey Eccentricity from beam xz-plane to nodal line. (+ve in the +ve local y direction). (See Notes)

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Rigidities:	RIGIDITIES 6 (Rigidities: Beam)
Matrix	Not applicable	
Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic		MATERIAL PROPERTIES NONLINEAR 29 (Elastic: Isotropic, Plastic: Resultant) (ifcode=1 or 2, see <i>Notes</i>)
Creep	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable	
Generic Polymer	Not applicable	
Composite	Not applicable	

Loading

Prescribed PDSP, Prescribed variable. U, V, W, θ_x , θ_y , θ_z : at nodes.

	Value	TPDSP	
Concentrated Loads	CL		Concentrated loads in global directions. Px, Py, Pz, Mx, My, Mz: at nodes.
Element Loads	ELDS		Element loads on nodal line (load type number LTYPE *10 defines the corresponding element load type on beam axis, see Note). LTYPE, S1, Px, Py, Pz, Mx, My, Mz LTYPE=11: point loads and moments in local directions. LTYPE=12: point loads and moments in global directions. LTYPE, 0, Wx, Wy, Wz, Mx, 0, 0 LTYPE=21: uniformly distributed loads in local directions. LTYPE=22: uniformly distributed loads in global directions (Mx=0). LTYPE=23: uniformly distributed projected loads in global directions (Mx=0). LTYPE, S1, Wx1, Wy1, Wz1, Mx1, 0, 0, S2, Wx2, Wy2, Wz2, Mx2, 0, 0 LTYPE=31: distributed loads in local directions. LTYPE=32: distributed loads in global directions (Mx1=0, Mx2=0). LTYPE=33: distributed projected loads in global directions (Mx1=0, Mx2=0). LTYPE, S1, Wx, Wy, Wz, Mx, 0, 0 LTYPE=41: trapezoidal loads in local directions. LTYPE=42: trapezoidal loads in global directions (Mx=0). LTYPE=43: trapezoidal projected loads in global directions (Mx=0).
Distributed Loads	UDL		Uniformly distributed loads. Wx, Wy, Wz: forces/unit length for element in local directions. (see <i>Notes</i>)
	FLD		Not applicable.
Body Forces	CBF		Constant body forces for Element. Xcbf, Ycbf, Zcbf, Ω_x , Ω_y , Ω_z , α_x , α_y , α_z
	BFP, BFPE		Not applicable.
Velocities	VELO		Velocities. Vx, Vy, Vz: at nodes.
Accelerations	ACCE		Acceleration. Ax, Ay, Az: at nodes
Initial	SSI, SSIE,		Initial stresses/strains at nodes/for element. Fx, Fy,

Stress/Strains	SSIG	Fz, Mx, My, Mz: axial force, shear forces, torque and moments in local directions. $\epsilon_x, \epsilon_y, \epsilon_z, \psi_x, \psi_y, \psi_z$: axial, shear and flexural strains in local directions.
Residual Stresses	SSR, SSRE, SSRG	Residual stresses at Gauss points. Resultants (for material model 29). Fx, Fy, Fz, Mx, My, Mz: axial force, shear forces, torque and moments in local directions.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, dT/dy, dT/dz, To, 0, dTo/dx, dTo/dz
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver Force: Fx, Fy, Fz, Mx, My, Mz: in local directions.
 Strain: $\epsilon_x, \epsilon_y, \epsilon_z, \psi_x, \psi_y, \psi_z$: Axial and shear strains, torsion and curvatures.
 By default element output is with respect to the nodal line. OPTION 418 outputs stress/strain resultants with respect to the beam centreline.

Modeller See [Results Tables \(Appendix K\)](#).

Local Axes

☐ [Standard line element](#)

Sign Convention

☐ [Standard beam element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	Not applicable.
Updated Lagrangian	Not applicable.
Eulerian	Not applicable.
Co-rotational	For large displacements and large rotations.

Integration Schemes

Stiffness Default. 1 point.
Fine. As default.
Mass Default. 1 point.
Fine. As default.

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 55 Output strains as well as stresses.
- 36 Follower loads
- 105 Lumped mass matrix.
- 157 Material model 29 (non cross-section elements), see *Notes*.
- 229 Co-rotational geometric nonlinearity.
- 102 Switch off load correction stiffness due to centripetal acceleration.
- 380 Output stress/strain resultants with respect to the beam centroidal axes (see *Notes*).
- 381 Input beam geometric properties and apply CBF, UDL, SSI and TEMP loads along beam centroidal axes
- 405 Specify geometric properties along beam centroidal axes
- 406 Specify CBF, UDL, SSI, SSR and TEMP loads along beam centroidal axes
- 414 Introduce residual bending flexibility correction for 2-node thick beam BTS3 (see *Notes*)
- 418 Output stress resultants relative to beam centroidal axes for eccentric elements (see *Notes*)

Notes on Use

1. The geometric properties can be input in the old format: A, Iyy, Izz, KT, Asz, Asy, ez (7 parameters), or A, Iyy, Izz, KT, Asz, Asy, ez, ey (8 parameters), or A, Iyy, Izz, KT, Asz, Asy, ez, ey, Iyz (9 parameters).
2. All forces and moments are constant along the length of the beam.
3. Displacement and rotation variations along the beam are linear.
4. OPTION 418 will replace OPTION 380. If OPTION 380 set TRUE in the dat file OPTION 418 will automatically be set TRUE within Solver.
5. Input of geometric properties (OPTION 405) and loads (OPTION 406), and output of stress/strain resultants (OPTION 418) are with respect to the beam

centroidal axis. CL is always input with respect to the nodal line; displacements are output with respect to the nodal line.

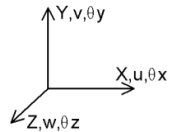
6. When BTS3 is used together with OPTION 414 to introduce residual bending flexibility correction, its stiffness matrix is enhanced to the order of a cubic (equivalent to BMS3). Note that if OPTION 414 is used with eccentrically stacked elements, slippage can occur.
7. This element is recommended for use in geometrically nonlinear applications involving large displacements and rotations but small strains, this would include 3D beam, frame or arch structures. They can be also used to model cables in cable-stayed structures. For linear problems more efficient and accurate solutions will be obtained using the BMS3 element as a result of the higher order interpolation function model.
8. The end releases for this element allow a joint to be modelled between adjacent elements. These joints allow rotation and translation of one beam with respect to another without load transferral. The rotations and translations remain in the local directions of the beam elements and support large deformations.
9. For nonlinear material model 29 the following geometric properties are appended to those already specified (see Geometric Properties).
 - A^P , Z_{yy}^P , Z_{zz}^P , Z_y^P , Z_z^P , S^P
 - A^P Plastic area (=elastic area)
 - Z_{yy}^P , Z_{zz}^P Plastic moduli for bending about y, z axes
 - Z_y^P , Z_z^P Plastic moduli for torsion about y, z axes.
 - S^P Plastic area for shear ($S^P=0$).
 - Where the fully plastic torsional moment = $\sigma_y (Z_y^P + Z_z^P)$.
 - Note that if eccentricity has been specified the plastic properties must be defined with reference to the **nodal line** and **not** the beam axes, i.e. the eccentricity is not used to automatically modify the plastic properties, they must be defined via modified geometry.
10. For nonlinear material model 29 the following **ifcode** parameters are applicable: **ifcode=1** for circular hollow sections and **ifcode=2** for solid rectangular sections.
11. Temperature dependent properties cannot be used with material model 29.
12. OPTION 36 is only applicable for use with element load types ELDS and UDL. Specifying this option makes these element loads follow the element geometry as the analysis progresses.

13. When a nonlinear material is used with this element the transverse shear stresses are excluded from the plasticity computations i.e. the transverse shear stresses are assumed to remain elastic. This means that if a nonlinear material is used in applications where transverse shear tends to dominate the stress field the equivalent von Mises and maximum principal stresses can exceed the uniaxial yield stress.
14. Specifying any geometric nonlinear option will result in the co-rotational formulation being used for BTS3 elements. This allows more flexibility of choice when other element types are included in the model.

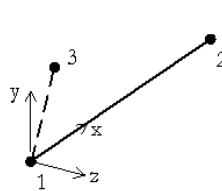
3D Thick Beam Elements

General

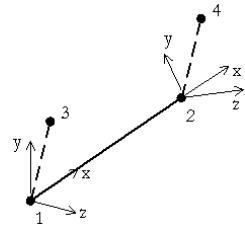
Element Name



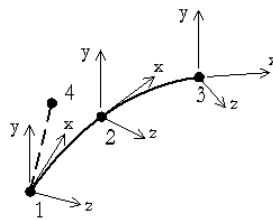
BMI21



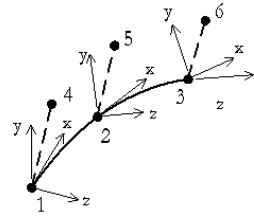
BMI22



BMI31



BMI33



Element Group

Beams

Element Subgroup

Thick Beams

Element Description

Straight and curved isoparametric degenerate thick beam elements in 3D for which shearing deformations are included. The elements can accommodate varying geometric properties along the length. BMI22 and BMI33 can consider initial twist.

Number Of Nodes

3 (BMI21), 4 (BMI22 and BMI31) and 6 (BMI33) with end release conditions.

Freeloms

$U, V, W, \theta_x, \theta_y, \theta_z$: at each active node.

End Releases

The element node numbers should be followed by: R restrained (default), F free defined in the order $U, V, W, \theta_x, \theta_y, \theta_z$ at node 1 and then $U, V, W, \theta_x, \theta_y, \theta_z$ at node 2 and node 3 (only for BMI31 and BMI33) related to local element axes (see Assumptions and Limitations).

Node Coordinates

X, Y, Z : at each node.

Geometric Properties

A, Iyy, Izz, Jxx,	At each node
Asz, Asy, Iyz, ez,	
ey	
A	Cross sectional area.
Iyy, Izz	2nd moment of area about local y, z directions (see Definition).
Jxx	Torsional constant . If input as zero, Iyy and Izz will be used to define the torsional properties (see the LUSAS Theory Manual)
Asz, Asy	Effective shear areas on local yz plane in local z, y directions (see shear areas).
Iy, Iz	1st moment of area about local y, z directions (see Definition).
Iyz	Product moment of area about local y, z axes (see Definition).
ez	Eccentricity from beam xy-plane to nodal line. (+ve in the +ve local z direction). (See Notes)
ey	Eccentricity from beam xz-plane to nodal line. (+ve in the +ve local y direction). (See Notes)

Note: For MATERIAL MODEL 29 additional geometric properties are appended to the previous 22 (BMI21 and BMI22) or 33 (BMI31 and BMI33) geometric properties (see Assumptions and Limitations).

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Rigidities:	RIGIDITIES 6 (Rigidities: Beam)
Matrix	Not applicable	
Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic	Stress resultant:	MATERIAL PROPERTIES NONLINEAR 29 (Elastic: Isotropic, Plastic: Resultant) (ifcode=1 or 2, see Assumptions and Limitations)
Creep	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)

Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable	
Generic Polymer	Not applicable	
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W, θ_x , θ_y , θ_z : at active nodes.
Concentrated Loads	CL	Concentrated loads in global directions. Px, Py, Pz, Mx, My, Mz: at active nodes.
Element Loads	ELDS	Element loads on nodal line (load type number LTYPE *10 defines the corresponding element load type on beam axis, see Assumptions and Limitations) (see Assumptions and Limitations) LTYPE, S1, Px, Py, Pz, Mx, My, Mz LTYPE=11: point loads and moments in local directions. LTYPE=12: point loads and moments in global directions. LTYPE, 0, Wx, Wy, Wz, Mx, My, Mz LTYPE=21: uniformly distributed loads in local directions. LTYPE=22: uniformly distributed loads in global directions (Mx=0). LTYPE=23: uniformly distributed projected loads in global directions. LTYPE, S1, Wx1, Wy1, Wz1, Mx1, My1, Mz1, S2, Wx2, Wy2, Wz2, Mx2, My2, Mz2 LTYPE=31: distributed loads in local directions. LTYPE=32: distributed loads in global directions. LTYPE=33: distributed projected loads in global directions. LTYPE, S1, Wx, Wy, Wz, Mx, My, Mz

		LTYPE=41: trapezoidal loads in local directions.
		LTYPE=42: trapezoidal loads in global directions.
		LTYPE=43: trapezoidal projected loads in global directions.
Distributed Loads	UDL	Uniformly distributed loads. Wx, Wy, Wz, Mx, My, Mz: local forces and moments / unit length for element (see Assumptions and Limitations).
	FLD	Not applicable.
Body Forces	CBF	Constant body forces for Element. Xcbf, Ycbf, Zcbf, Ω_x , Ω_y , Ω_z , α_x , α_y , α_z
	BFP, BFPE	Body force potentials at nodes/for element. ϕ_1 , ϕ_2 , ϕ_3 , 0, Xcbf, Ycbf, Zcbf
Velocities	VELO	Velocities. Vx, Vy, Vz: at nodes.
Accelerations	ACCE	Acceleration. Ax, Ay, Az: at nodes
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. Fx, Fy, Fz, Mx, My, Mz: axial force, shear forces, torque and moments in local directions. ϵ_x , ϵ_y , ϵ_z , ψ_x , ψ_y , ψ_z : axial, shear and flexural strains in local directions.
	SSIG	Initial stresses/strains at Gauss points. These stresses/strains are specified in the same manner as SSI and SSIE.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. Resultants (for material model 29). Fx, Fy, Fz, Mx, My, Mz: axial force, shear forces, torque and moments in local directions.
	SSRG	Residual stresses at Gauss points. These stresses are specified in the same manner as SSR and SSRE.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, dT/dy, dT/dz, To, 0, dTo/dy, dTo/dz in local directions
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Stress resultants (default): Fx, Fy, Fz, Mx, My, Mz: axial force, shear forces, torque and moments in local directions.

Strain: $\epsilon_x, \epsilon_y, \epsilon_z, \psi_x, \psi_y, \psi_z$: Axial, shear, torsional and flexural strains in local directions.

By default element output is with respect to the nodal line. OPTION 418 outputs stress/strain resultants with respect to the beam centroidal axes.

Modeller See [Results Tables \(Appendix K\)](#).

Local Axes

- ☐ [Standard line element](#) For each element/active node, the local xy-plane is defined by the local x-axis and the orientation node. The local y-axis is perpendicular to the local x-axis and positive on the side of the element where the orientation node lies. The local y and z-axes form a right-handed set with the local x-axis. See [Local Element Axes](#) for details

Sign Convention

- ☐ [Standard beam element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian Not applicable.
Updated Lagrangian Not applicable.
Eulerian Not applicable.
Co-rotational Not applicable.

Integration Schemes

Stiffness Default. 1-point (BMI21 and BMI22), 2-point (BMI31 and BMI33).
Fine. Same as default.
Mass Default. 2-point (BMI21 and BMI22), 3-point (BMI31 and BMI33).
Fine. Same as default.

Note: A 3-point [Newton-Cotes integration](#) rule is also available for BMI31 and BMI33 using OPTION 134. This may be more applicable for infinitesimal strain, elasto-plastic analyses of plane frames with straight members since the first and third quadrature points will coincide with the frame joints. See Appendix I of the Theory Manual.

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 36 Follower loads
- 55 Output strains as well as stresses.
- 102 Switch off load correction stiffness matrix due to centripetal acceleration
- 105 Lumped mass matrix.
- 134 Gauss to [Newton-Cotes](#) in plane (in the local x direction) integration for elements
- 157 Material model 29 (non cross-section elements), see Notes.
- 403 Introduce residual bending flexibility correction for 2-node thick beam BMI21, see Assumptions and Limitations.
- 404 Compute equivalent nodal loading from equilibrium considerations for 2-node thick beam BMI21, see Assumptions and Limitations.
- 405 Specify geometric properties along beam centroidal axes
- 406 Specify CBF, UDL, SSI, SSR and TEMP loads along beam centroidal axes
- 418 Output stress resultants relative to beam centroidal axes for eccentric elements

Notes, Assumptions and Limitations

1. The element is formulated from the so-called degenerate continuum concept, i.e. enforcing directly the modified Timoshenko hypothesis for thick beams to the continuum theory. Plane cross-sections initially normal to the beam axis remain plane and undistorted (the shape of the cross-section remains unchanged) under deformation, but do not necessarily remain normal to the beam axis; the shear centre and centroid of cross-section coincide. Shearing deformations are included. The basic kinematic assumptions correspond to the Timoshenko beam theory and do not allow for warping effects in torsion. Although warping effects can be considered approximately by using real torsional constants, inaccuracies are likely to occur when eccentricity is present.
2. Input of geometric properties (OPTION 405) and loads (OPTION 406), and output of stress/strain resultants (OPTION 418) are with respect to the beam centroidal axes. CL is always input with respect to the nodal line; displacements are output with respect to the nodal line.
3. The axial force, shear forces, moments and torsion are constant in BMI21 and BMI22, and vary linearly along the length of the beam in BMI31 and BMI33.
4. When BMI21 is used together with OPTION 403 to introduce residual bending flexibility correction, its stiffness matrix is enhanced to the order of a cubic (equivalent to BMS3). Note that if OPTION 403 is used with eccentrically stacked elements, slippage can occur.
5. When BMI21 is used together with OPTION 404, loading that varies along the element length is accounted for in the force diagrams (i.e. for a beam under CBF or internal element loading). A post-processing technique has been introduced to obtain accurate quadratic bending moments for BMI31. For BMI21 (with OPTION 404) and BMI31, internal forces and moments are

- output at intervals of 1/10th of the element length by specifying the Gauss point option from the Output button of the LUSAS Datafile dialog.
6. The end releases for this element allow a joint to be modelled between adjacent elements. These joints allow rotation and translation of one beam with respect to another without load transferral. The rotations and translations remain in the local directions of the beam elements and support large deformations.
 7. For nonlinear material model 29 the following geometric properties are appended to those already specified (see Geometric Properties).
 - A^P , Z_{yy}^P , Z_{zz}^P , Z_y^P , Z_z^P , S^P at each node
 - A^P Plastic area (=elastic area)
 - Z_{yy}^P , Z_{zz}^P Plastic moduli for bending about y, z axes
 - Z_y^P , Z_z^P Plastic moduli for torsion about y, z axes.
 - S^P Plastic area for shear ($S^P=0$).

Where the fully plastic torsional moment = $\sigma_y (Z_y^P + Z_z^P)$.

Note that if eccentricity has been specified the plastic properties must be defined with reference to the **nodal line** and **not** the beam axes, i.e. the eccentricity is not used to automatically modify the plastic properties, they must be defined via modified geometry.

For nonlinear material model 29 the following **ifcode** parameters are applicable: **ifcode=1** for circular hollow sections and **ifcode=2** for solid rectangular sections.

8. Temperature dependent properties cannot be used with material model 29.
9. The [rigidity matrix](#) is evaluated explicitly from the geometric properties for both linear and nonlinear materials.
10. OPTION 36 is only applicable for use with element load types ELDS and UDL. Specifying this option makes these element loads follow the element geometry as the analysis progresses.
11. When a nonlinear material is used with this element the transverse shear stresses are excluded from the plasticity computations i.e. the transverse shear stresses are assumed to remain elastic. This means that if a nonlinear material is used in applications where transverse shear tends to dominate the stress field the equivalent von Mises and maximum principal stresses can exceed the uniaxial yield stress.
12. If a step by step dynamic analysis is carried out (with damping) using BMI21/BMI22 elements with distributed loading, the force diagrams would not be correct until all inertia forces have been damped out and the beam reaches a state of static equilibrium. For this reason, the “free body force diagrams” pertaining to applied loading, are not included in the analysis. This means that different force diagrams will now be obtained for BMI21/BMI22 elements if

static and dynamic analyses are directly compared. Therefore, in a dynamic analysis, BMI21/BMI22 elements will perform in the same manner as other beam elements in LUSAS where free body force diagrams are never included.

Restrictions

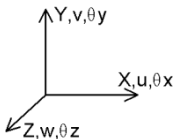
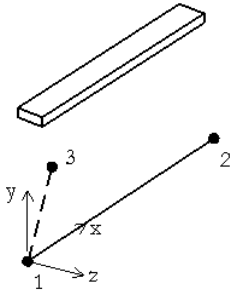
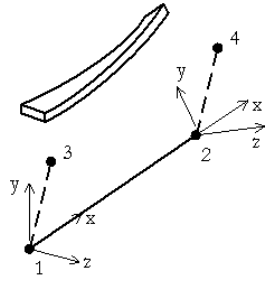
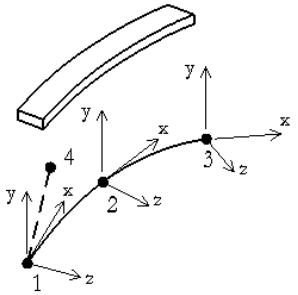
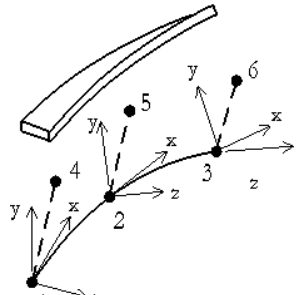
- ☐ [Ensure mid-side node centrality](#)
- ☐ Avoid excessive element curvature
- ☐ BMI22 and BMI33 are not available for selection currently within LUSAS Modeller.

Recommendations on Use

- The elements may be used for linear and material nonlinear analysis of three dimensional beam, frame and arch structures. BMI21 and BMI22 may also be used as a stiffener for the QTS4 shell element; while BMI31 and BMI33 may be used as a stiffener for the QTS8 shell element.
- When options 403 and 404 are true, BMI21 element is identical to the 2-noded straight beam (BMS3) for linear analysis of structures containing straight members of constant cross-section, e.g. space frames.

3D Thick Beam Elements with Quadrilateral Cross-Section

General

Element Name	BMX21	BMX22
		
Element Name	BMX31	BMX33
		
Element Group	Beams	
Element Subgroup	Thick Beams	
Element Description	Straight and curved isoparametric degenerate thick beam elements in 3D for which shearing deformations are included. The element has a quadrilateral cross section which may vary along the element length. BMX22 and BMX33 can consider initial twist.	
Number Of Nodes	3 (BMX21), 4 (BMX22 and BMX31) and 6 (BMX33) with end release conditions.	
Freedoms	U, V, W, θ_x , θ_y , θ_z : at each active node.	
End Releases	The element node numbers should be followed by: R restrained (default), F free defined in the order U, V, W, θ_x , θ_y , θ_z at node 1	

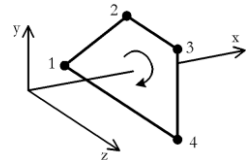
**Node
Coordinates**

and then $U, V, W, \theta_x, \theta_y, \theta_z$ at node 2 and node 3 (only for BMX31 and BMX33) related to local element axes (see Notes).
 X, Y, Z : at each node.

Geometric Properties

$y_1, z_1, y_2, z_2, y_3, z_3, y_4, z_4$: local cross section coordinate pairs at each node; followed by nt12, nt14: number of [Newton-Cotes](#) integration points in the direction defined by the local cross-section points 1-2 and 1-4 (zero indicates default values). Multiple quadrilateral cross-sections can be used to build up complex beam cross-sections.

Note. The corners of the quadrilateral are numbered clockwise about the local x-axis (the beam nodal line), that is, a right-hand screw rule in the direction of increasing x.



Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
Matrix	Not applicable	
Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic	Stress resultant:	Not applicable.
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	Not applicable.
	Stress Potential	STRESS POTENTIAL VON_MISES (Isotropic: von Mises, Modified von Mises)
Creep	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP

		(Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic	Not applicable	
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable	
Generic	Not applicable	
Polymer		
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W, θ_x , θ_y , θ_z : at active nodes.
Concentrated Loads	CL	Concentrated loads in global directions. Px, Py, Pz, Mx, My, Mz: at active nodes (global).
Element Loads	ELDS	<u>Element loads</u> on nodal line (load type number LTYPE *10 defines the corresponding element load type on beam axis, see Assumptions and Limitations) LTYPE, S1, Px, Py, Pz, Mx, My, Mz LTYPE=11: point loads and moments in local directions. LTYPE=12: point loads and moments in global directions. LTYPE, 0, Wx, Wy, Wz, Mx, My, Mz LTYPE=21: uniformly distributed loads in local directions. LTYPE=22: uniformly distributed loads in global directions (Mx=0). LTYPE=23: uniformly distributed projected loads in global directions. LTYPE, S1, Wx1, Wy1, Wz1, Mx1, My1, Mz1, S2, Wx2, Wy2, Wz2, Mx2, My2, Mz2 LTYPE=31: distributed loads in local

		directions.
		LTYPE=32: distributed loads in global directions.
		LTYPE=33: distributed projected loads in global directions.
		LTYPE, S1, Wx, Wy, Wz, Mx, My, Mz
		LTYPE=41: trapezoidal loads in local directions.
		LTYPE=42: trapezoidal loads in global directions.
		LTYPE=43: trapezoidal projected loads in global directions.
Distributed Loads	UDL	Uniformly distributed loads. Wx, Wy, Wz, Mx, My, Mz: local forces and moments / unit length for element in local directions. See Assumptions and Limitations.
	FLD	Not applicable.
Body Forces	CBF	Constant body forces for Element. Xcbf, Ycbf, Zcbf, Ω_x , Ω_y , Ω_z , α_x , α_y , α_z
	BFP, BFPE	Body force potentials at nodes/for element. ϕ_1 , ϕ_2 , ϕ_3 , 0, Xcbf, Ycbf, Zcbf
Velocities	VELO	Velocities. Vx, Vy, Vz: at nodes.
Accelerations	ACCE	Acceleration. Ax, Ay, Az: at nodes
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. Components: 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, (σ_x , 0, 0, ϵ_x , 0, 0) Bracketed terms repeated for each fibre integration point.
	SSIG	Initial stresses/strains at Gauss points. These stresses/strains are specified in the same manner as SSI and SSIE.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. Components: 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, (σ_x , 0, 0) Bracketed terms repeated for each fibre integration point.
	SSRG	Residual stresses at Gauss points. These stresses are specified in the same manner as SSR and SSRE.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, dT/dy, dT/dz, To, 0, dTo/dy, dTo/dz in local directions
Field Loads	Not applicable.	
Temp Dependent	Not applicable.	

Loads

LUSAS Output

Solver Stress resultants (default): F_x , F_y , F_z , M_x , M_y , M_z : axial force, shear forces, torque and moments in local directions.
Continuum stresses (OPTION 172): σ_x , σ_{xy} , σ_{xz} : in local directions.
Strain: ϵ_x , ϵ_y , ϵ_z , ψ_x , ψ_y , ψ_z : Axial, shear, torsional and flexural strains in local directions.
Continuum strains (OPTION 172): ϵ_x , ϵ_{xy} , ϵ_{xz} : in local directions.
By default element output is with respect to the nodal line. OPTION 418 outputs stress/strain resultants with respect to the beam centroidal axes.

Modeller See [Results Tables \(Appendix K\)](#).

Local Axes

- ❑ [Standard line element](#) For each element/active node, the local xy-plane is defined by the local x-axis and the orientation node. The local y-axis is perpendicular to the local x-axis and positive on the side of the element where the orientation node lies. The local y and z-axes form a right-handed set with the local x-axis. See [Local Element Axes](#) for details

Sign Convention

- ❑ [Standard beam element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	Not applicable.
Updated Lagrangian	Not applicable.
Eulerian	Not applicable.
Co-rotational	Not applicable.

Integration Schemes

Stiffness Default. 1-point (BMX21 and BMX22), 2-point (BMX31 and BMX33).
Fine. Same as default.
Mass Default. 2-point (BMX21 and BMX22), 3-point (BMX31 and BMX33).
Fine. Same as default.

Note: A 3-point [Newton-Cotes integration](#) rule is also available for BMX31 and BMX33 using OPTION 134. This may be more applicable for infinitesimal strain, elasto-plastic analyses of plane frames with straight members since the first and third

quadrature points will coincide with the frame joints. See Appendix I of the Theory Manual.

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 36 Follower loads
- 55 Output strains as well as stresses.
- 102 Switch off load correction stiffness matrix due to centripetal acceleration
- 105 Lumped mass matrix.
- 134 Gauss to [Newton-Cotes](#) in plane (in the local x direction) integration for elements.
- 139 Output yielded integration points only.
- 172 Form the [rigidity matrix](#) by numerical cross section integration.
- 404 Compute equivalent nodal loading from equilibrium considerations for 2-node thick beam BMX21, see Notes.
- 406 Specify CBF, UDL, SSI, SSR and TEMP loads along beam centroidal axes
- 418 Output stress resultants relative to beam centroidal axes for eccentric elements

Notes, Assumptions and Limitations

1. The element is formulated from the so-called degenerate continuum concept, i.e. enforcing directly the modified Timoshenko hypothesis for thick beams to the continuum theory. Plane cross-sections initially normal to the beam axis remain plane and undistorted (the shape of the cross-section remains unchanged) under deformation, but do not necessarily remain normal to the beam axis; the shear centre and centroid of cross-section coincide. Shearing deformations are included.
2. Input of loads (OPTION 406) and output of stress/strain resultants (OPTION 418) are with respect to the beam centroidal axis. CL is always input with respect to the nodal line; displacements are output with respect to the nodal line. Fiber stress/strain results are output at the actual location.
3. The axial force, shear forces, moments and torsion are constant in BMX21 and BMX22, and vary linearly along the length of the beam in BMX31 and BMX33.
4. When BMX21 is used together with OPTION 404, loading that varies along the element length is accounted for in the force diagrams (i.e. for a beam under CBF or internal element loading). Internal forces and moments are output at intervals of 1/10th of the element length by specifying the Gauss point option from the Output button of the LUSAS Datafile dialog.
5. The end releases for this element allow a joint to be modelled between adjacent elements. These joints allow rotation and translation of one beam with respect to

another without load transferral. The rotations and translations remain in the local directions of the beam elements and support large deformations.

6. Computation of the rigidity matrix by integration through the cross-section depth of the beam is necessary for all nonlinear material models. By default OPTION 172 is invoked automatically and a 5*5 point Newton-Cotes integration rule is used. This allows the output of stresses at the numerical cross section integration points.
7. By default, the rigidity matrix is evaluated explicitly for linear materials. A 3*3 point Newton-Cotes integration rule may be invoked using OPTION 172. Numerical cross section integration enables top, middle and bottom stress output.
8. OPTION 36 is only applicable for use with element load types ELDS and UDL. Specifying this option makes these element loads follow the element geometry as the analysis progresses.
9. The torsional constant is estimated from the computed values for Iyy and Izz, $J_{xx} = I_{yy} + I_{zz}$.
10. For nonlinear material models, fibre integration is used across the cross-sectional area of the beam. Only axial deformation is considered in the plasticity computations, any torsional deformation is assumed to remain elastic.

Restrictions

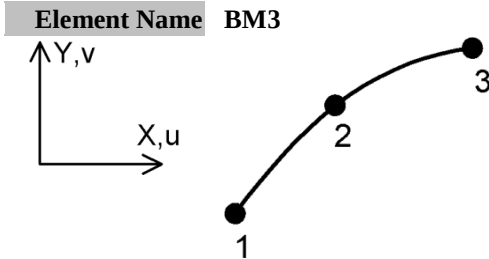
- ☐ [Ensure mid-side node centrality](#)
- ☐ Avoid excessive element curvature
- ☐ BMX22 and BMX33 are not available for selection currently within LUSAS Modeller.

Recommendations on Use

- The elements may be used for linear and nonlinear analysis of three dimensional beam, frame and arch structures. BMX21 and BMX22 may also be used as a stiffener for the QTS4 shell element; while BMX31 and BMX33 may be used as a stiffener for the QTS8 shell element.

2D Kirchhoff Thin Beam Elements

General



Element Group	Beams
Element Subgroup	Kirchhoff Beams
Element Description	Parabolically curved thin beam element in which shear deformations are excluded. The element can accommodate varying geometric properties along the length.
Number Of Nodes	3
Freedom	U, V, θ_z : at end nodes. dU: (relative displacement) at mid-side node.
Node Coordinates	X, Y: at each node.

Geometric Properties

- A, Izz, ey** At each node
- A** Cross sectional area
 - Izz** 2nd moment of area about local z-axis (see [Definition](#)).
 - ey** Eccentricity from beam xz-plane to nodal line (+ve in +ve local y-direction)

For a beam with [eccentricity](#) e from the nodal line then $I_{zz}=e^2A+I_{na}$ and $I_z=eA$ ($I_{na}=I$ about centroidal axis).

For MATERIAL MODEL 29 additional geometric properties are appended to the previous 9 geometric properties; see *Notes*.

Material Properties

- Linear** .. Isotropic: MATERIAL PROPERTIES (Elastic: Isotropic)
- Rigidities: RIGIDITIES 3 (Rigidities:Beam)
- Matrix** Not applicable

Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic	Stress resultant:	MATERIAL PROPERTIES NONLINEAR 29 (Elastic: Isotropic, Plastic: Resultant) (ifcode=1 or 2, see <i>Notes</i>)
Creep	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable	
Generic	Not applicable	
Polymer		
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, θ_z : at end nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py, Mz: at end nodes. dPx: in local x direction at mid-side node.
Element Loads	ELDS	Element loads on nodal line (load type number LTYPE *10 defines the corresponding element load type on beam axis). LTYPE, S1, Px, Py, Mz LTYPE=11: point loads and moments in local directions. LTYPE=12: point loads and moments in global directions. LTYPE, 0, Wx, Wy, Mz LTYPE=21: uniformly distributed loads in local directions. LTYPE=22: uniformly distributed loads in global directions.

		LTYPE=23: uniformly distributed projected loads in global directions
		LTYPE, S1, Wx1, Wy1, Mz1, S2, Wx2, Wy2, Mz2
		LTYPE=31: distributed loads in local directions.
		LTYPE=32: distributed loads in global directions.
		LTYPE=33: distributed projected loads in global directions
		LTYPE, S1, Wx, Wy, Mz
		LTYPE=41: trapezoidal loads in local directions.
		LTYPE=42: trapezoidal loads in global directions.
		LTYPE=43: trapezoidal projected loads in global directions
Distributed Loads	UDL	Uniformly distributed loads. Wx, Wy: force/unit length in local directions.
	FLD	Not applicable.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, $\Omega_x, \Omega_y, \Omega_z, \alpha_z$
	BFP, BFPE	Body force potentials at nodes/for element. $\phi_1, \phi_2, 0, 0, Xcbf, Ycbf$
Velocities	VELO	Velocities. Vx, Vy: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay: at nodes
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. Fx, Mz, 0: forces, moments in local directions. $\epsilon_x, \psi/z, 0$: strains in local directions.
	SSIG	Initial stresses/strains at Gauss points Fx, Mz, 0: forces, moments in local directions. $\epsilon_x, \psi/z, 0$: strains in local directions.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. Fx, Mz, 0: forces, moments in local directions.
	SSRG	Residual stresses at Gauss points Fx, Mz, 0: forces, moments in local directions.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, dT/dy, 0, To, 0, dTo/dy, 0
Field Loads	Not applicable.	
Temp	Not applicable.	
Dependent Loads		

LUSAS Output

Solver Force (default): Fx, Fy, Mz: forces, moments in local directions (see *Notes*).

Strain: ϵ_x , ϵ_y , ψ_z : axial, flexural strains in local directions.

By default element output is with respect to the nodal line. OPTION 418 outputs stress/strain resultants with respect to the beam centroidal axis.

Modeller See [Results Tables \(Appendix K\)](#).

Local Axes

- ☐ [Standard line element](#)

Sign Convention

- ☐ [Standard beam element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian For large displacements, small rotations and small strains.

Updated Lagrangian For large displacements, large rotations and small strains.

Eulerian Not applicable.

Co-rotational Not applicable.

Integration Schemes

Stiffness Default. 2-point.
Fine (see *Options*). 3-point.

Mass Default. 2-point.
Fine (see *Options*). 3-point.

A 3-point [Newton-Cotes integration](#) rule is also available using option 134. This may be more applicable for infinitesimal strain, elastoplastic analyses of plane frames with straight members since the first and third quadrature points will coincide with the frame joints. See Appendix I of the Theory manual.

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes fine integration rule for element.
- 54** Updated Lagrangian geometric nonlinearity.
- 55** Output strains as well as stresses.
- 87** Total Lagrangian geometric nonlinearity.
- 105** Lumped mass matrix
- 134** Gauss to [Newton-Cotes](#) in plane (in the local x direction) integration for elements.
- 157** Material model 29 (non cross-section elements), see *Notes*.
- 170** Suppress transfer of shape function arrays to disk.
- 405** Specify geometric properties along beam centroidal axes
- 406** Specify CBF, UDL, SSI, SSR and TEMP loads along beam centroidal axes.
- 418** Output stress resultants relative to beam centroidal axes for eccentric elements

Notes on Use

1. The element formulation is based on the constrained super-parametric approach. The variation of axial force along the beam is linear. The variation of displacement is quadratic in the local x-direction and cubic in the local y-direction. Shear force is constant.
2. Input of geometric properties (OPTION 405) and loads (OPTION 406), and output of stress/strain resultants (OPTION 418) are with respect to the beam centroidal axis. CL is always input with respect to the nodal line; displacements are output with respect to the nodal line.
3. For nonlinear material model 29 the following geometric properties are appended to those already specified (see *Geometric Properties*).
 - A^P , Z_{zz}^P , S^P at each node (i.e. nodes 1, 2, 3)
 - A^P Plastic area (=elastic area)
 - Z_{zz}^P Plastic modulus for bending about z axis
 - S^P Plastic area for shear ($S^P=0$)
3. For nonlinear material model 29 the following ifcode parameters should be
 - ifcode=1 for circular hollow sections.
 - ifcode=2 for solid rectangular sections.

4. Temperature dependent properties cannot be used with material model 29.
5. The element should not be coupled to the face of a two dimensional continuum element because of the midside node incompatibility.
6. The [rigidity matrix](#) for BM3 is evaluated explicitly from the material and geometric properties for both linear and nonlinear materials.

Restrictions

- ☐ [Ensure mid-side node centrality](#)
- ☐ Avoid excessive element curvature

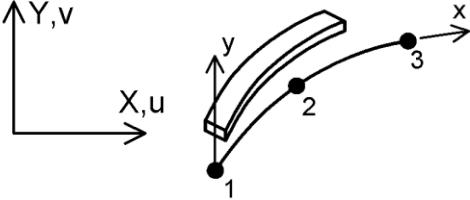
Recommendations on Use

The element may be used for linear and nonlinear analysis of two dimensional beam, frame and arch structures. The 2-noded straight beam (BEAM) is more effective for the linear analysis of structures containing straight members of constant cross-section, e.g. plane frames.

2D Kirchhoff Thin Beam Element with Quadrilateral Cross-Section

General

Element Name	BMX3

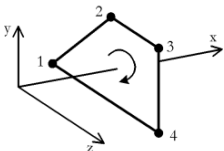


Element Group	Beams
Element Subgroup	Kirchhoff Beams
Element Description	Parabolically curved thin beam elements in which shear deformations are excluded. The quadrilateral cross-section may be eccentric and can vary along the element length.
Number Of Nodes	3
Freedom	U, V, θ_z : at end nodes. dU: (relative displacement) at mid-side node.
Node Coordinates	X, Y: at each node.

Geometric Properties

$y_1, z_1, y_2, z_2, y_3, z_3, y_4, z_4$: local cross section coordinate pairs at each node; followed by nt12, nt14: specifying the number of [Newton-Cotes integration](#) points in the direction defined by the local cross-section points 1-2 and 1-4 (zero indicates default values). See *Notes*. Multiple quadrilateral cross-sections can be used to build up complex beam cross-sections.

Note. The coordinates of the cross section are numbered clockwise about the local x-axis (the beam nodal line). That is, a right-hand screw rule in the direction of increasing x.



Material Properties

Linear Isotropic: MATERIAL PROPERTIES (Elastic: Isotropic)

Matrix	Not applicable	
Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic	Stress resultant:	MATERIAL PROPERTIES NONLINEAR 29 (Elastic: Isotropic, Plastic: Resultant) (ifcode=2, see <i>Notes</i>)
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	Not applicable
	Stress Potential	STRESS POTENTIAL VON_MISES (Isotropic: von Mises, Modified von Mises)
Creep		CREEP PROPERTIES (Creep)
	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic	Not applicable	
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable	
Generic Polymer	Not applicable	
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, θ_z : at end nodes. dU at mid-side node.
Concentrated Loads	CL	Concentrated loads. Px, Py, Mz: at end nodes (global). dPx: at mid-side node (local).
Element Loads	ELDS	Element loads on nodal line (load type number LTYPE *10 defines the corresponding element load type on beam axis). LTYPE, S1, Px, Py, Mz LTYPE=11: point loads and moments in local directions. LTYPE=12: point loads and moments in global directions. LTYPE, 0, Wx, Wy, Mz LTYPE=21: uniformly distributed loads in local directions. LTYPE=22: uniformly distributed loads in global directions. LTYPE=23: uniformly distributed projected loads in global directions LTYPE, S1, Wx1, Wy1, Mz1, S2, Wx2, Wy2, Mz2 LTYPE=31: distributed loads in local directions. LTYPE=32: distributed loads in global directions. LTYPE=33: distributed projected loads in global directions LTYPE, S1, Wx, Wy, Mz LTYPE=41: trapezoidal loads in local directions. LTYPE=42: trapezoidal loads in global directions. LTYPE=43: trapezoidal projected loads in global directions
Distributed Loads	UDL	Uniformly distributed loads. Wx, Wy: force/unit length in local directions.
	FLD	Not applicable.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Ω_x , Ω_y , Ω_z , α_z
	BFP, BFPE	Body force potentials at nodes/for element. ϕ_1 , ϕ_2 , 0, 0, Xcbf, Ycbf
Velocities	VELO	Velocities. Vx, Vy: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay: at nodes
Initial	SSI, SSIE	Initial stresses/strains at nodes/for element.

Stress/Strains		Resultants (for linear material models without numerical cross section integration and model 29, see <i>Notes</i>): F_x , M_z , 0: forces, moments in local directions. ϵ_x , ψ/z , 0: strains in local directions.
	SSIG	Initial stresses/strains at Gauss points. (1) Resultants (for linear material models without numerical cross section integration and model 29, see <i>Notes</i>). F_x , M_z , 0: forces, moments in local directions. ϵ_x , ψ/z , 0 strains in local directions. (2) Components (for linear material models with numerical cross section integration and all non-linear material models except 29): 0, 0, 0, 0, 0, 0, (σ_x, ϵ_x) . Bracketed terms repeated at each fibre integration point.
Residual Stresses	SSR, SSRE	Not applicable.
	SSRG	Residual stresses at Gauss points. (1) Resultants (material model 29): F_x , M_z , 0 (2) Components (all nonlinear material models except 29, also linear material models with numerical cross section integration): 0, 0, 0, 0, 0, 0, (σ_x) Bracketed term repeated for each fibre integration point.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element T, 0, dT/dy , 0, T_0 , 0, dT_0/dy , 0: in local directions.
Field Loads	Not applicable.	
Temp	Not applicable.	
Dependent Loads		

LUSAS Output

Solver	Force (default): F_x , M_z , F_y : forces, moment in local directions (see <i>Notes</i>) Continuum stresses (OPTION 172): σ_x : in local directions. Strain: ϵ_x , ψ/z , 0 : axial, flexural strains in local directions. Continuum strains (OPTION 172): ϵ_x : in local directions. By default element output is with respect to the nodal line. OPTION 418 outputs stress/strain resultants with respect to the beam centroidal axis.
Modeller	See Results Tables (Appendix K) .

Local Axes

- ☐ [Standard line element](#)

Sign Convention

- ☐ [Standard beam element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	For large displacements, small rotations and small strains.
Updated Lagrangian	For large displacements, large rotations and small strains.
Eulerian	Not applicable.
Co-rotational	Not applicable.

Integration Schemes

Stiffness	Default.	2-point.
	Fine (see <i>Options</i>).	3-point.
Mass	Default.	2-point.
	Fine (see <i>Options</i>).	3-point.

A 3-point [Newton-Cotes integration](#) rule is also available using option 134. This may be more applicable for infinitesimal strain, elastoplastic analyses of plane frames with straight members since the first and third quadrature points will coincide with the frame joints. See Appendix I of the Theory manual.

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes fine integration rule for element.
- 32** Suppress stress output but not resultants
- 54** Updated Lagrangian geometric nonlinearity.
- 55** Output strains as well as stresses
- 87** Total Lagrangian geometric nonlinearity
- 105** Lumped mass matrix
- 134** Gauss to [Newton-Cotes](#) in plane (in the local x direction) integration for elements.
- 157** Material model 29 (non cross-section elements), see *Notes*.

- 170 Suppress transfer of shape function arrays to disk.
- 172 Formulate [rigidity matrix](#) by integrating across the cross-section
- 406 Specify CBF, UDL, SSI, SSR and TEMP loads along beam centroidal axes
- 418 Output stress resultants relative to beam centroidal axes for eccentric elements

Notes on Use

1. The element formulation is based on the constrained super-parametric approach. The variation of axial force along the beam is linear. The variation of displacement is quadratic in the local x-direction and cubic in the local y-direction. Shear force is constant.
2. Input of loads (OPTION 406), and output of stress/strain resultants (OPTION 418) are with respect to the beam centroidal axis. CL is always input with respect to the nodal line; displacements are output with respect to the nodal line. Fiber stress/strain results are output at the actual location.
3. Initial strain resultants may be input for any material model.
4. The number of numerical cross-section integration points, nt12 and nt14, may be specified but for improved performance the number of integration points corresponding to the y direction can be defined correctly (the beam bends about the local z-axis) and the integration rule in the other direction may be set to 1.
5. For nonlinear material model 29 ifcode must be set to 2 for solid rectangular sections. Multiple quadrilateral cross-sections can be used to build up complex beam cross-sections.
6. Temperature dependent properties cannot be used with material model 29.
7. The element should not be coupled to the face of a two dimensional continuum element because of the midside node incompatibility.
8. Computing the [rigidity matrix](#) by integration through the cross-section depth of the beam is necessary for all nonlinear material models (except 29). By default option 172 is invoked automatically and a 5 point [Newton-Cotes integration](#) rule is used.
9. By default, the [rigidity matrix](#) is evaluated explicitly for linear materials. A 3 point [Newton-Cotes](#) rule may be invoked using option 172. Numerical cross section integration enables top, middle and bottom stress output.

Restrictions

- ☐ [Ensure mid-side node centrality](#)
- ☐ Avoid excessive element curvature

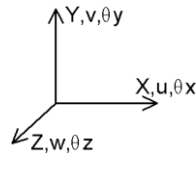
Recommendations on Use

The element may be used for linear and nonlinear analysis of two dimensional beam, frame and arch structures. The 2-noded straight beam (BEAM) is more effective for linear analysis of structures containing straight members of constant cross-section, e.g. plane frames.

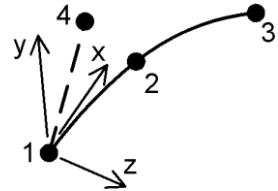
3D Kirchhoff Thin Beam Elements

General

Element Name BS3



BS4



Element Group

Beams

Element

[Kirchhoff](#) Beams

Subgroup

Element

Description

Curved beam elements in 3D for which shearing deformations are excluded. The elements can accommodate varying geometric properties along the length.

Number Of

Nodes

3 (BS3).

4 (BS4). The 4th node is used to define the local xy-plane.

Freedom

U, V, W, θ_x , θ_y , θ_z : at end nodes (1 and 3)

dU, d θ_x : (relative displacement/rotation) at mid-length node.

Node

Coordinates

X, Y, Z: at each node.

Geometric Properties

A, Iyy, Izz, Jxx, At each node

Iy, Iz, Iyz, ez, ey

A Cross sectional area

Iyy, Izz 2nd moment of area about local y, z directions (see [Definition](#))

Jxx [Torsional constant](#). If input as zero, Iyy and Izz will be used to define the torsional properties (see the *LUSAS Theory Manual*)

Iy, Iz 1st moment of area about local y, z directions (see [Definition](#))

Iyz Product moment of area (see [Definition](#))

ez Eccentricity from beam xy-plane to nodal line. (+ve in the +ve local z direction). (See Notes)

ey Eccentricity from beam xz-plane to nodal line. (+ve in the +ve local y direction). (See Notes)

For MATERIAL MODEL 29 additional geometric properties are appended to the previous 21 geometric properties (see *Notes*).

Material Properties

Linear	Isotropic: Rigidities:	MATERIAL PROPERTIES (Elastic: Isotropic) RIGIDITIES 6 (Rigidities: Beam)
Matrix	Not applicable	
Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic	Stress resultant:	MATERIAL PROPERTIES NONLINEAR 29 (Elastic: Isotropic, Plastic: Resultant) (ifcode=1 or 2, see <i>Notes</i>)
Creep	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable	
Generic	Not applicable	
Polymer		
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W, θ_x , θ_y , θ_z : at end nodes (1 and 3). dU, d θ_x : at mid-length node.
Concentrated Loads	CL	Concentrated loads. Px, Py, Pz, Mx, My, Mz: at end nodes. dPx, dMy: at mid-length node.
Element Loads	ELDS	Element loads on nodal line (load type number LTYPE *10 defines the corresponding element load type on beam axis) LTYPE, S1, Px, Py, Pz, Mx, My, Mz LTYPE=11: point loads and moments in local directions. LTYPE=12: point loads and moments in global

		directions.
		LTYPE, 0, Wx, Wy, Wz, Mx, My, Mz
		LTYPE=21: uniformly distributed loads in local directions.
		LTYPE=22: uniformly distributed loads in global directions.
		LTYPE=23: uniformly distributed projected loads in global directions.
		LTYPE, S1, Wx1, Wy1, Wz1, Mx1, My1, Mz1, S2, Wx2, Wy2, Wz2, Mx2, My2, Mz2
		LTYPE=31: distributed loads in local directions.
		LTYPE=32: distributed loads in global directions.
		LTYPE=33: distributed projected loads in global directions.
		LTYPE, S1, Wx, Wy, Wz, Mx, My, Mz
		LTYPE=41: trapezoidal loads in local directions.
		LTYPE=42: trapezoidal loads in global directions.
		LTYPE=43: trapezoidal projected loads in global directions.
Distributed Loads	UDL	Uniformly distributed loads. Wx, Wy, Wz: local forces/unit length.
	FLD	Not applicable.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf, Ω_x , Ω_y , Ω_z
	BFP, BFPE	Body force potentials at nodes/for element. ϕ_1 , ϕ_2 , ϕ_3 , 0, Xcbf, Ycbf, Zcbf
Velocities	VELO	Velocities. Vx, Vy, Vz: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay, Az: at nodes
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. Fx, My, Mz, Txz, Txy, 0: axial force, moments and torques in local directions. ϵ_x , ψ_y , ψ_z , ψ_{xz} , ψ_{xy} , 0: axial, flexural and torsional strains in local directions. Total torque = Txz + Txy, total torsional strain = $y_{xz} + \psi_{xy}$.
	SSIG	Not applicable.
Residual Stresses	SSR, SSRE	Not applicable.
	SSRG	Residual stresses at Gauss points. Resultants (for material model 29). Fx, My, Mz, Txz, Txy, 0: axial force, moments and torques in local directions. Total torque = Txz + Txy, total torsional strain = $y_{xz} + \psi_{xy}$.

Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, dT/dy, dT/dz, To, 0, dTo/dy, dTo/dz
Field Loads	Not applicable.	
Temp	Not applicable.	
Dependent Loads		

LUSAS Output

Solver	Force (default): Fx, Fy, Fz, My, Mz, Txz, Txy: axial force, moments, torques and shear forces in local directions. (Total torque = Txz+Txy). Strain: ϵ_x , ψ_y , ψ_z , ψ_{xz} , ψ_{xy} , 0: axial, flexural and torsional strains in local directions. By default element output is with respect to the nodal line. OPTION 418 outputs stress/strain resultants with respect to the beam centroidal axes.
Modeller	See Results Tables (Appendix K) .

Local Axes

For BS3 the local xy-plane is defined by the 3 element nodes. The local y-axis is perpendicular to the local x-axis and positive on the convex side of the element. The local y and z-axes form a right handed set with the local x-axis.

For BS4 the local xy-plane is defined by the 2 end nodes of the beam and the 4th node. The local y-axis is perpendicular to the local x-axis and positive on the side of the element where the 4th node lies. The local y and z-axes form a right handed set with the local x-axis. See [Local Element Axes](#) for more details.

Sign Convention

- ☐ [Standard beam element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	For large displacements, small rotations and small strains.
Updated Lagrangian	For large displacements, large rotations and small strains.
Eulerian	Not applicable.
Co-rotational	Not applicable.

Integration Schemes

Stiffness	Default.	2-point.
	Fine (see <i>Options</i>).	3-point.

Mass Default.	2-point.
Fine (see Options).	3-point.

A 3-point [Newton-Cotes](#) integration rule is also available using option 134. This may be more applicable for infinitesimal strain, elastoplastic analyses of plane frames with straight members since the first and third quadrature points will coincide with the frame joints. See Appendix I of the Theory manual.

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes fine integration rule for element.
- 54** Updated Lagrangian geometric nonlinearity.
- 55** Output strains as well as stresses.
- 87** Total Lagrangian geometric nonlinearity.
- 102** Switch off load correction stiffness matrix due to centripetal acceleration.
- 105** Lumped mass matrix.
- 134** Gauss to [Newton-Cotes](#) in plane (in the local x direction) integration for elements.
- 157** Material model 29 (non cross-section elements), see *Notes*.
- 170** Suppress transfer of shape function arrays to disk.
- 405** Specify geometric properties along beam centroidal axes
- 406** Specify CBF, UDL, SSI, SSR and TEMP loads along beam centroidal axes
- 418** Output stress resultants relative to beam centroidal axes for eccentric elements

Notes on Use

1. The element formulation is based on the [Kirchhoff](#) hypothesis for thin beams (i.e. the exclusion of shearing deformations).
2. The variation of axial force, moments and torsion along the length of the bar can be regarded as linear. Shear force variations are constant.
3. Input of geometric properties (OPTION 405) and loads (OPTION 406), and output of stress/strain resultants (OPTION 418) are with respect to the beam centroidal axes. CL is always input with respect to the nodal line; displacements are output with respect to the nodal line.
4. For nonlinear material model 29 the following geometric properties are appended to those already specified (see *Geometric Properties*).

- A^p , Z_{yy}^p , Z_{zz}^p , Z_y^p , Z_z^p , S^p at each node (i.e. nodes 1, 2, 3).
- A^p Plastic area (=elastic area)
- Z_{yy}^p , Z_{zz}^p Plastic moduli for bending about y, z axes
- Z_y^p , Z_z^p Plastic moduli for torsion about y, z axes.
- S^p Plastic area for shear ($S^p=0$).

Where the fully plastic torsional moment = $\sigma_y (Z_y^p + Z_z^p)$.

For nonlinear material model 29 the following ifcode parameters should be

- ifcode=1 for circular hollow sections.
 - ifcode=2 for solid rectangular sections.
4. Temperature dependent properties cannot be used with material model 29.
 5. The element should not be coupled to the edges of either continuum or shell elements because of midside node incompatibility.
 6. The [rigidity matrix](#) for BS3 and BS4 is evaluated explicitly from the geometric properties for both linear and nonlinear materials.

Restrictions

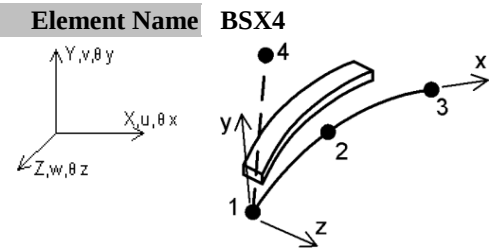
- ☐ [Ensure mid-side node centrality](#)
- ☐ Avoid excessive element curvature

Recommendations on Use

The elements may be used for linear and nonlinear analysis of three dimensional beam, frame and arch structures. The 2-noded straight beam (BMS3) is more effective for linear analysis of structures containing straight members of constant cross-section, e.g. space frames.

3D Kirchhoff Thin Beam Element with Quadrilateral Cross-Section

General

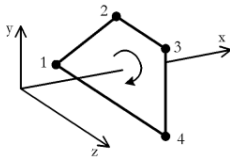


Element Name	BSX4
Element Group	Beams
Element Subgroup	Kirchhoff Beams
Element Description	Curved beam elements in 3D for which shearing deformations are excluded. The element has a quadrilateral cross section which may vary along the element length.
Number Of Nodes	4. The 4th node is used to define the local xy-plane.
Freedom	U, V, W, θ_x , θ_y , θ_z : at the end nodes (1 and 3) dU, d θ_x : (relative displacement/rotation) at the mid-length node.
Node Coordinates	X, Y, Z: at each node.

Geometric Properties

y1, z1, y2, z2, y3, z3, y4, z4: local cross section coordinate pairs at each node; followed by nt12, nt14: specifying the number of [Newton-Cotes](#) integration points in the direction defined by the local cross-section points 1-2 and 1-4 (zero indicates default values). Multiple quadrilateral cross-sections can be used to build up complex beam cross-sections.

Note. The coordinates of the cross section are numbered clockwise about the local x-axis (the beam nodal line). That is, a right-hand screw rule in the direction of increasing x.



Material Properties

Linear Isotropic: MATERIAL PROPERTIES (Elastic: Isotropic)

Matrix	Not applicable	
Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic	Stress resultant:	Not applicable.
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	Not applicable
	Stress Potential	STRESS POTENTIAL VON_MISES (Isotropic: von Mises, Modified von Mises)
Creep		CREEP PROPERTIES (Creep)
	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic Shrinkage	Not applicable	
		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable	
Generic Polymer	Not applicable	
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W, θ_x , θ_y , θ_z : at the end nodes. dU, d θ_x : at the mid-length node.
Concentrated Loads	CL	Concentrated loads. Px, Py, Pz, Mx, My, Mz: at end nodes (global). dPx, dMx: at mid-length local node.
Element Loads	ELDS	Element loads on nodal line (load type number LTYPE *10 defines the corresponding element load type on beam axis) LTYPE, S1, Px, Py, Pz, Mx, My, Mz LTYPE=11: point loads and moments in local directions. LTYPE=12: point loads and moments in global directions. LTYPE, 0, Wx, Wy, Wz, Mx, My, Mz LTYPE=21: uniformly distributed loads in local directions. LTYPE=22: uniformly distributed loads in global directions. LTYPE=23: uniformly distributed projected loads in global directions. LTYPE, S1, Wx1, Wy1, Wz1, Mx1, My1, Mz1, S2, Wx2, Wy2, Wz2, Mx2, My2, Mz2 LTYPE=31: distributed loads in local directions. LTYPE=32: distributed loads in global directions. LTYPE=33: distributed projected loads in global directions. LTYPE, S1, Wx, Wy, Wz, Mx, My, Mz LTYPE=41: trapezoidal loads in local directions. LTYPE=42: trapezoidal loads in global directions. LTYPE=43: trapezoidal projected loads in global directions.
Distributed Loads	UDL	Uniformly distributed loads. Wx, Wy, Wz: forces/unit length in local directions.
	FLD	Not applicable
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf, Ω_x , Ω_y , Ω_z , α_x , α_y , α_z
	BFP, BFPE	Body force potentials at nodes/for element. ϕ_1 , ϕ_2 , ϕ_3 , 0, Xcbf, Ycbf, Zcbf
Velocities	VELO	Velocities. Vx, Vy, Vz: at nodes.

Accelerations	ACCE	Acceleration Ax, Ay, Az: at nodes
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. Components: 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, (σ_x , σ_{xy} , σ_{xz} , σ_{yz} , ε_{yz} , ε_x , ε_{xz} , ε_{yz}) Bracketed terms repeated for each fibre integration point.
	SSIG	Initial stresses/strains at Gauss points. These stresses/strains are specified in the same manner as SSI and SSIE.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. Components: 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, (σ_x , σ_{xy} , σ_{xz} , σ_{yz}) Bracketed terms repeated for each fibre integration point.
	SSRG	Residual stresses at Gauss points. These stresses are specified in the same manner as SSR and SSRE.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, dT/dy, dT/dz, To, 0, dTo/dy, dTo/dz: in local directions.
Field Loads	Not applicable	
Temp Dependent Loads	Not applicable	

LUSAS Output

Solver	Force (default): Fx, My, Mz, Txz, Txy, Fy, Fz: axial force, moments, torques and shear forces in local directions. (Total Torque = Txz + Txy). Continuum stresses (OPTION 172): σ_x , σ_{xy} , σ_{xz} , σ_{yz} : in local directions. Strain: ε_x , ψ_y , ψ_z , ψ_{xz} , ψ_{xy} : axial, flexural and torsional strains in local directions. Continuum strains (OPTION 172): ε_x , ε_{xy} , ε_{xz} , ε_{yz} : in local directions. By default element output is with respect to the nodal line. OPTION 418 outputs stress/strain resultants with respect to the beam centroidal axes.
Modeller	See Results Tables (Appendix K) .

Local Axes

Standard line element. The local xy-plane is defined by the 2 end nodes of the beam and the 4th node. The local y-axis is perpendicular to the x-axis and positive on the side of the element where the 4th node lies.

The local y and z-axes form a right-hand set with the local x-axis.

Sign Convention

- ☐ [Standard beam element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	For large displacements, small rotations and small strains.
Updated Lagrangian	For large displacements, large rotations and small strains.
Eulerian	Not applicable.
Co-rotational	Not applicable.

Integration Schemes

Stiffness	Default.	2-point.
	Fine (see <i>Options</i>).	3-point.
Mass	Default.	2-point.
	Fine (see <i>Options</i>).	3-point.

A 3-point [Newton-Cotes](#) integration rule is also available using option 134. This may be more applicable for infinitesimal strain, elastoplastic analyses of plane frames with straight members since the first and third quadrature points will coincide with the frame joints. See Appendix I of the Theory manual.

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes fine integration rule for element.
- 54** Updated Lagrangian geometric nonlinearity.
- 55** Output strains as well as stresses.
- 87** Total Lagrangian geometric nonlinearity.
- 102** Switch off load correction stiffness matrix due to centripetal acceleration.
- 105** Lumped mass matrix.
- 134** Gauss to [Newton-Cotes](#) in plane (in the local x direction) integration for elements.
- 139** Output yielded integration points only.
- 170** Suppress transfer of shape function arrays to disk.
- 172** Form the [rigidity matrix](#) by numerical cross section integration.
- 406** Specify CBF, UDL, SSI, SSR and TEMP loads along beam centroidal axes

418 Output stress resultants relative to beam centroidal axes for eccentric elements

Notes on Use

1. The element formulation is based on the [Kirchhoff](#) hypothesis for thin beams (i.e. the exclusion of shearing deformations)
2. The variation of axial force, moments and torsion along the length of the bar can be regarded as linear. Shear force is constant.
3. Input of loads (OPTION 406), and output of stress/strain resultants (OPTION 418) are with respect to the beam centroidal axes. CL is always input with respect to the nodal line; displacements are output with respect to the nodal line. Fiber stress/strain results are output at their actual location.
4. Computation of the [rigidity matrix](#) by integration over the thickness is necessary for all nonlinear material models. For nonlinear models a 5x5 [Newton-Cotes integration](#) rule is used as default. For linear models a 3x3 rule is used as the default. This allows the output of stresses at the numerical cross section integration points.
5. The torsional constant is estimated from the computed values for I_{yy} and I_{zz} , $J_{xx} = I_{yy} + I_{zz}$.
6. For nonlinear material models, fibre integration is used across the cross-sectional area of the beam. Only axial deformation is considered in the plasticity computations, any torsional deformation is assumed to remain elastic.
7. The element should not be coupled to the face of a two dimensional continuum element because of the midside node incompatibility.
8. Computing the [rigidity matrix](#) by integration through the cross-section depth of the beam is necessary for all nonlinear material models (except 29). By default OPTION 172 is invoked automatically and a 5*5 point [Newton-Cotes integration](#) rule is used.
9. By default, the [rigidity matrix](#) is evaluated explicitly for linear materials. A 3*3 point [Newton-Cotes integration](#) rule may be invoked using OPTION 172. Numerical cross section integration enables top, middle and bottom stress output.

Restrictions

- ☐ [Ensure mid-side node centrality](#)
- ☐ Avoid excessive element curvature

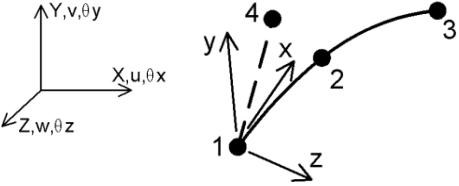
Recommendations on Use

The elements may be used for linear and nonlinear analysis of three dimensional beam, frame and arch structures. The 2-noded straight beam (BMS3) is more effective for

linear analysis of structures containing straight members of constant cross-section, e.g. space frames.

3D Semiloof Thin Beam Elements

General

Element Name	BSL3, BSL4
	
Element Group	Beams
Element Subgroup	Semiloof Beams
Element Description	Curved beam elements in 3D which can be mixed with the semiloof shell elements TSL6 and QSL8. The elements can accommodate varying geometric properties. Shearing deformations are excluded.
Number Of Nodes	3 or 4. For BSL4 the 4th node is used to define the local xy-plane.
Freedom	U, V, W, θ_x , θ_y , θ_z : at end nodes (1 and 3). U, V, W, θ_1 , θ_2 : at mid-side node (node 2) (see <i>Notes</i>).
Node Coordinates	X, Y, Z: at each node.

Geometric Properties

A, Iyy, Izz, Jxx, at nodes 1, 2 and 3

Iy, Iz, Iyz, ez, ey

A Cross sectional area

Iyy, Izz 2nd moments of area in local y, z axes (see [Definition](#))

Jxx [Torsional constant](#). If input as zero, Iyy and Izz will be used to define the torsional properties (see the *LUSAS Theory Manual*)

Iy, Iz 1st moment of area in local y, z axes (see [Definition](#))

Iyz Product moment of area (see [Definition](#)).

ez Eccentricity from beam xy-plane to nodal line (+ve in +ve local z-direction)

ey Eccentricity from beam xz-plane to nodal line (+ve in +ve local y-direction)

For MATERIAL MODEL 29 additional geometric properties are appended to the 21 properties above; see *Notes*.

Material Properties

Linear	Isotropic: Rigidities:	MATERIAL PROPERTIES (Elastic: Isotropic) RIGIDITIES Rigidities 6 (Rigidities: Beam)
Matrix	Not applicable	
Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic	Stress resultant:	MATERIAL PROPERTIES NONLINEAR 29 (Elastic: Isotropic, Plastic: Resultant) (ifcode=1 or 2, see <i>Notes</i>)
Creep	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable	
Generic	Not applicable	
Polymer		
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W, θ_x , θ_y , θ_z : at end nodes. U, V, W, θ_1 , θ_2 : at mid-side node.
Concentrated Loads	CL	Concentrated loads. Px, Py, Pz, Mx, My, Mz: at end nodes (global). Px, Py, Pz, M1, M2: at mid-side node (M1 and M2 local).
Element Loads	ELDS	Element loads on nodal line (load type number LTYPE *10 defines the corresponding element load type on beam axis) LTYPE, S1, Px, Py, Pz, Mx, My, Mz LTYPE=11: point loads and moments in local directions.

		LTYP=12: point loads and moments in global directions. LTYP=0, W_x, W_y, W_z, M_x, M_y, M_z LTYP=21: uniformly distributed loads in local directions. LTYP=22: uniformly distributed loads in global directions. LTYP=23: uniformly distributed projected loads in global directions. LTYP, S1, W_{x1}, W_{y1}, W_{z1}, M_{x1}, M_{y1}, M_{z1}, S2, W_{x2}, W_{y2}, W_{z2}, M_{x2}, M_{y2}, M_{z2} LTYP=31: distributed loads in local directions. LTYP=32: distributed loads in global directions. LTYP=33: distributed projected loads in global directions. LTYP, S1, W_x, W_y, W_z, M_x, M_y, M_z LTYP=41: trapezoidal loads in local directions. LTYP=42: trapezoidal loads in global directions. LTYP=43: trapezoidal projected loads in global directions.
Distributed Loads	UDL	Uniformly distributed loads. W _x , W _y , W _z : force/unit length in local directions for element.
	FLD	Not applicable.
Body Forces	CBF	Constant body forces for element. X _{cbf} , Y _{cbf} , Z _{cbf} , Ω _x , Ω _y , Ω _z , α _x , α _y , α _z
	BFP, BFPE	Body force potentials at nodes/for element. φ ₁ , φ ₂ , φ ₃ , 0, X _{cbf} , Y _{cbf} , Z _{cbf}
Velocities	VELO	Velocities. V _x , V _y , V _z : at nodes.
Accelerations	ACCE	Accelerations. A _x , A _y , A _z : at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. F _x , M _y , M _z , T _{xz} , T _{xy} , 0 in local directions. ε _x , ψ _y , ψ _z , ψ _{xz} , ψ _{xy} , 0: in local directions. (see <i>Notes</i>). Total torque = T _{xz} + T _{xy}
	SSIG	Not applicable.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. Resultants (nonlinear model 29): F _x , M _y , M _z , T _{xz} , T _{xy} , 0: in local directions.
	SSRG	Not applicable.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, dT/dy, dT/dz, T _o , 0, dT _o /dy, dT _o /dz: in local

directions.

Field Loads Not applicable.
Temp Not applicable.
Dependent Loads

LUSAS Output

Solver Force (default): F_x , M_y , M_z , T_{xz} , T_{xy} , F_y , F_z : in local directions.
(Total torque = $T_{xz} + T_{xy}$)
Strain: ϵ_x , ψ_y , ψ_z , ψ_{xz} , ψ_{xy} : in local directions. (see *Notes*). Total
torsional strain = $\psi_{xz} + \psi_{xy}$
By default element output is with respect to the nodal line. OPTION 418
outputs stress/strain resultants with respect to the beam centroidal axes.

Modeller See [Results Tables \(Appendix K\)](#).

Local Axes

[Standard line element](#). For BSL3 the local xy-plane is defined by the 3 element nodes. The local y-axis is perpendicular to the local x-axis and positive on the convex side of the element. The local y and z-axes form a right-hand set with the local x-axis. For BSL4 the local xy-plane is defined by the 2 end nodes of the beam and the 4th node. The local y-axis is perpendicular to the x-axis and positive on the side of the element where the 4th node lies. The local y and z-axes form a right-hand set with the local x-axis.

Sign Convention

☐ [Standard beam element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian For large displacements, small rotations and small strains.
Updated Lagrangian Not applicable.
Eulerian Not applicable.
Co-rotational Not applicable.

Integration Schemes

Stiffness Default. 3-point torsion, 2-point bending.
Fine. As default.
Mass Default. 3-point.

Fine. As default.

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 55 Output strains as well as stresses.
- 87 Total Lagrangian geometric nonlinearity
- 102 Switch off load correction stiffness matrix due to centripetal acceleration.
- 105 Lumped mass matrix.
- 157 Material model 29 (non cross-section elements), see *Notes*.
- 170 Suppress transfer of shape function arrays to disk.
- 405 Specify geometric properties along beam centroidal axes
- 406 Specify CBF, UDL, SSI, SSR and TEMP loads along beam centroidal axes
- 418 Output stress resultants relative to beam centroidal axes for eccentric elements

Notes on Use

1. The semiloof beam element is based on a [Kirchhoff](#) hypothesis for thin beams (i.e. the exclusion of shearing deformations).
2. The variation of axial force, moments and torsion can be regarded as linear along the length of the element. Shear forces are constant along the length of the element.
3. The loof rotations θ_1 and θ_2 refer to rotations about the element at the loof positions. A positive loof rotation is defined by a right-hand screw rule applied to a vector running in the local x-axis direction along the element edge.
4. Input of geometric properties (OPTION 405) and loads (OPTION 406), and output of stress/strain resultants (OPTION 418) are with respect to the beam centroidal axes. CL is always input with respect to the nodal line; displacements are output with respect to the nodal line.
5. For nonlinear material model 29 the following geometric properties are appended to those already specified (see *Geometric Properties*).
 - A^p , Z_{yy}^p , Z_{zz}^p , Z_y^p , Z_z^p , S^p at each node (i.e. nodes 1, 2, 3).
 - A^p Plastic area (=elastic area)
 - Z_{yy}^p , Z_{zz}^p Plastic moduli for bending about y, z axes
 - Z_y^p , Z_z^p Plastic moduli for torsion about y, z axes.

- S^p Plastic area for shear ($S^p=0$).

Where the fully plastic torsional moment = $\sigma_y (Z_y^p + Z_z^p)$

5. For nonlinear material model 29 the following ifcode parameters should be
 - ifcode=1 for circular hollow sections.
 - ifcode=2 for solid rectangular sections.
6. Semiloof beam elements should be used with semiloof shell elements. For beam only problems, BS3/BS4 elements should be used.
7. Temperature dependent properties cannot be used with material model 29.
8. Integration of the element stiffness matrix is performed using selective integration, with a 2-point Gauss rule for the axial and flexural strain energy, and a 3-point Gauss rule for the torsional strain energy. The selective integration technique is implemented in a similar manner to the method proposed by Hughes [H4], i.e. the strain-displacement matrix for the bending and axial strains is evaluated at the reduced rule quadrature points and then extrapolated to the sampling locations of the 3-point quadrature rule. The material response is then assessed at the 3-point Gauss rule.
9. The [rigidity matrix](#) for BSL3 and BSL4 is evaluated explicitly from the geometric properties for both linear and nonlinear materials.

Restrictions

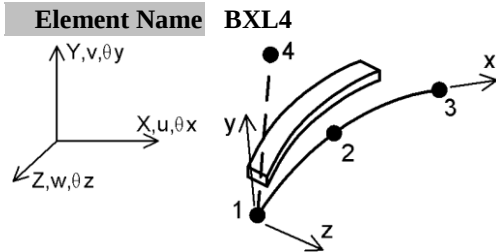
- ☐ [Ensure mid-side node centrality](#)
- ☐ Avoid excessive element curvature

Recommendations on Use

- The primary use of this element is to provide a beam stiffener for the semiloof shell (QSL8) for analysing stiffened shell structures.
- The BS3 and BS4 elements are more effective for linear analysis of 3D frame structures with curved members and nonlinear analysis of three dimensional beam, frame and arch structures.
- The 2-noded straight beam (BMS3) is the most effective for linear analysis of structures containing straight members of constant cross-section, e.g. space frames.

3D Semiloof Thin Beam Element with Quadrilateral Cross-Section

General

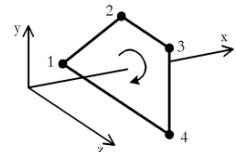


Element Name	BXL4
Element Group	Beams
Element Subgroup	Semiloof Beams
Element Description	A curved beam element in 3D which can be mixed with the semiloof shell element. The element has a quadrilateral cross section which may vary along the element. Shearing deformations are excluded.
Number Of Nodes	4. The 4th node is used to define the local xy-plane.
Freedom	U, V, W, θ_x , θ_y , θ_z : at end nodes. U, V, W, θ_1 , θ_2 : at mid-length node.
Node Coordinates	X, Y, Z: at each node.

Geometric Properties

$y_1, z_1, y_2, z_2, y_3, z_3, y_4, z_4$: local cross section coordinate pairs at each node; followed by nt12, nt14: number of [Newton-Cotes](#) integration points in the direction defined by the local cross-section points 1-2 and 1-4 (zero indicates default values). Multiple quadrilateral cross-sections can be used to build up complex beam cross-sections.

Note. The corners of the quadrilateral are numbered clockwise about the local x-axis (the beam nodal line), that is, a right-hand screw rule in the direction of increasing x.



Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
Matrix	Not applicable	

Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic	Stress resultant:	Not applicable.
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	Not applicable
	Stress Potential	STRESS POTENTIAL VON_MISES (Isotropic: von Mises, Modified von Mises)
Creep		CREEP PROPERTIES (Creep)
	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic	Not applicable	
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable	
Generic Polymer	Not applicable	
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W, θ_x , θ_y , θ_z : at end nodes. U, V, W, θ_1 , θ_2 at mid-side node.
Concentrated Loads	CL	Concentrated loads Px, Py, Pz, Mx, My, Mz at end nodes (global). Px, Py, Pz, M1, M2: at mid-side node (M1 and M2 local).
Element Loads	ELDS	Element loads on nodal line (load type number LTYPE *10 defines the corresponding element load type on beam axis) LTYPE, S1, Px, Py, Pz, Mx, My, Mz LTYPE=11: point loads and moments in local directions. LTYPE=12: point loads and moments in global directions. LTYPE, 0, Wx, Wy, Wz, Mx, My, Mz LTYPE=21: uniformly distributed loads in local directions. LTYPE=22: uniformly distributed loads in global directions. LTYPE=23: uniformly distributed projected loads in global directions. LTYPE, S1, Wx1, Wy1, Wz1, Mx1, My1, Mz1, S2, Wx2, Wy2, Wz2, Mx2, My2, Mz2 LTYPE=31: distributed loads in local directions. LTYPE=32: distributed loads in global directions. LTYPE=33: distributed projected loads in global directions. LTYPE, S1, Wx, Wy, Wz, Mx, My, Mz LTYPE=41: trapezoidal loads in local directions. LTYPE=42: trapezoidal loads in global directions. LTYPE=43: trapezoidal projected loads in global directions.
Distributed Loads	UDL	Uniformly distributed loads. Wx, Wy, Wz: for element in local directions.
	FLD	Not applicable.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf, Ω_x , Ω_y , Ω_z , α_x , α_y , α_z
	BFP, BFPE	Body force potentials at nodes/for element. ϕ_1 , ϕ_2 , ϕ_3 , 0, Xcbf, Ycbf, Zcbf
Velocities	VELO	Velocities. Vx, Vy, Vz: at nodes.

Accelerations	ACCE	Accelerations. Ax, Ay, Az: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. Components: 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, (σ_x , σ_{xy} , σ_{xz} , σ_{yz} , ϵ_{yz} , ϵ_x , ϵ_{xz} , ϵ_{yz}) Bracketed terms repeated for each fibre integration point.
	SSIG	Initial stresses/strains at Gauss points. These stresses/strains are specified in the same manner as SSI and SSIE
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. Components: 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, (σ_x , σ_{xy} , σ_{xz} , σ_{yz}) Bracketed terms repeated for each fibre integration point.
	SSRG	Residual stresses at Gauss points. These stresses are specified in the same manner as SSR and SSRE.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, dT/dy, dT/dz, To, 0, dTo/dy, dTo/dz
Field Loads	Not applicable.	
Temp	Not applicable.	
Dependent Loads		

LUSAS Output

Solver	Force (default): Fx, My, Mz, Txz, Txy, Fy, Fz: in local directions. Total torque = Txz+Txy. Continuum stresses (Option 172): σ_x , σ_{xy} , σ_{xz} , σ_{yz} : in local directions. Strain/curvatures (default): ϵ_x , ψ_y , ψ_z , ψ_{xz} , ψ_{xy} , γ_{yz} : in local directions (see <i>Notes</i>). Total torsional strain = ψ_{xy} + ψ_{yz} . Continuum strains (Option 172): ϵ_x , ϵ_{xy} , ϵ_{xz} , ϵ_{yz} : in local directions. By default element output is with respect to the nodal line. OPTION 418 outputs stress/strain resultants with respect to the beam centroidal axes.
Modeller	See Results Tables (Appendix K) .

Local Axes

- **Standard line element** The local xy-plane is defined by the 2 end nodes of the beam and the 4th node. The local y-axis is perpendicular to the x-axis and positive on the side of the element where the 4th node lies. The local y and z-axes form a right-hand set with the local x-axis.

Sign Convention

- ☐ [Standard beam element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	For large displacements, large rotations and small strains.
Updated Lagrangian	Not applicable.
Eulerian	Not applicable.
Co-rotational	Not applicable.

Integration Schemes

Stiffness	Default.	2-point torsion, 2-point bending.
	Fine.	As default.
Mass	Default.	3-point.
	Fine.	As default.

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 32 Suppress stress output (but not stress resultant).
- 55 Output strains as well as stresses.
- 87 Total Lagrangian geometric nonlinearity.
- 102 Disable load correction stiffness matrix due to centripetal acceleration.
- 105 Lumped mass matrix
- 139 Output inelastic Gauss points only
- 170 Suppress transfer of shape function arrays to disk
- 172 Form the [rigidity matrix](#) by numerical cross section integration.
- 406 Specify CBF, UDL, SSI, SSR and TEMP loads along beam centroidal axes
- 418 Output stress resultants relative to beam centroidal axes for eccentric elements

Notes, Assumptions and Limitations

1. The semiloof beam element formulation is based on a [Kirchhoff](#) hypothesis for thin beams (i.e. shearing deformations are excluded). The variation of axial force, bending and torsion along the length of the element may be considered as linear. Shear forces are constant.

2. Input of loads (OPTION 406), and output of stress/strain resultants (OPTION 418) are with respect to the beam centroidal axes. CL is always input with respect to the nodal line; displacements are output with respect to the nodal line. Fiber stress/strain results are output at their actual location.
3. The torsional constant is estimated from the computed values for I_{yy} and I_{zz} , $J_{xx} = I_{yy} + I_{zz}$.
4. For nonlinear material models, fibre integration is used across the cross-sectional area of the beam. Only axial deformation is considered in the plasticity computations, any torsional deformation is assumed to remain elastic.
5. Computing the [rigidity matrix](#) by integration through the cross-section depth of the beam is necessary for all nonlinear material models (except 29). By default option 172 is invoked automatically and a 5*5 point [Newton-Cotes integration](#) rule is used.
6. By default, the [rigidity matrix](#) is evaluated explicitly for linear materials. A 3*3 point [Newton-Cotes integration](#) rule may be invoked using option 172. Numerical cross section integration enables top, middle and bottom stress output.
7. Integration of the element stiffness matrix is performed using selective integration, with a 2-point Gauss rule for the axial and flexural strain energy, and a 3-point Gauss rule for the torsional strain energy. The selective integration technique is implemented in a similar manner to the method proposed by Hughes, i.e. the strain-displacement matrix for the bending and axial strains is evaluated at the reduced rule quadrature points and then extrapolated to the sampling locations of the 3-point quadrature rule. The material response is then assessed at the 3-point Gauss rule.

Restrictions

- ☐ [Ensure mid-side node centrality](#)
- ☐ Avoid excessive element curvature

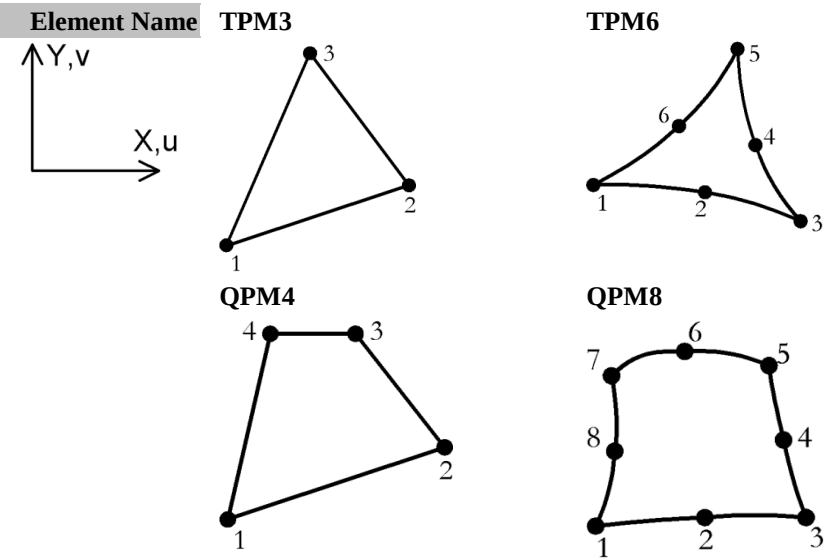
Recommendations on Use

- The element's primary use is to provide a beam stiffener for the semiloof shell (QSL8) for analysing stiffened shell structures.
- The BSX4 element is more effective for linear analysis of 3D frame structures with curved members and nonlinear analysis of three dimensional beam, frame and arch structures.
- The 2-noded straight beam (BMS3) is the most effective for linear analysis of structures containing straight members of constant cross-section, e.g. space frames.

Chapter 3 : 2D Continuum Elements.

2D Plane Stress Continuum Elements

General



Element Group	2D Continuum
Element Subgroup	Plane Stress Continuum
Element Description	A family of 2D isoparametric elements with the higher order elements capable of modelling curved boundaries. The elements are numerically integrated.
Number Of Nodes	3, 4, 6 or 8, numbered anticlockwise.
Freedom	U, V: at each node.
Node Coordinates	X, Y: at each node.

Geometric Properties

t1... tn Thickness at each node.

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC (Elastic: Orthotropic Plane Stress)

	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC 3 (Elastic: Anisotropic Thin Plate)
	Rigidities.	RIGIDITIES 3 (Rigidities: Membrane/Thin Plate)
Matrix	Not applicable	
Joint	Not applicable	
Concrete		MATERIAL PROPERTIES NONLINEAR 94 (Elastic: Isotropic, Plastic: Multi-Crack Concrete)
		MATERIAL PROPERTIES NONLINEAR 102 (Elastic: Isotropic, Plastic: Smoothed Multi-Crack Concrete)
Elasto-Plastic	Stress resultant:	Not applicable.
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Volumetric Crushing:	Not applicable
	Interface:	MATERIAL PROPERTIES NONLINEAR 27
	Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
Creep		CREEP PROPERTIES (Creep)
	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Chinese creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic	Not applicable	

Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable	
Generic Polymer	Isotropic	MATERIAL PROPERTIES NONLINEAR 89 (Generic Polymer Model)
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V: at nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py: at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	Face Loads . Px, Py: Local Face Axis Pressures At Nodes.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, $\Omega_x, \Omega_y, \Omega_z, \alpha_z$
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, ϕ_4 , Xcbf, Ycbf
Velocities	VELO	Velocities. Vx, Vy: at nodes.
Accelerations	ACCE	Accelerations. Ax, Ay: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. $\sigma_x, \sigma_y, \sigma_{xy}$: global stresses. $\epsilon_x, \epsilon_y, \gamma_{xy}$: global strains.
	SSIG	Initial stresses/strains at Gauss points. $\sigma_x, \sigma_y, \sigma_{xy}$: global stresses. $\epsilon_x, \epsilon_y, \gamma_{xy}$: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. $\sigma_x, \sigma_y, \sigma_{xy}$: global stresses.
	SSRG	Residual stresses at Gauss points. $\sigma_x, \sigma_y, \sigma_{xy}$: global stresses.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

- Solver** Stress resultants: N_x , N_y , N_{xy} , N_{max} , N_{min} , β , N_s , N_e
 Stress (default): σ_x , σ_y , σ_{xy} , σ_{max} , σ_{min} , β , σ_s , σ_e (see [description of principal stresses](#))
 Strain: ϵ_x , ϵ_y , γ_{xy} , ϵ_{max} , ϵ_{min} , β , ϵ_s , ϵ_e
- Modeller** See [Results Tables \(Appendix K\)](#).

Local Axes

Not applicable (global axes are the reference).

Sign Convention

- ☐ Standard 2D continuum element

Formulation

Geometric Nonlinearity

- Total Lagrangian** For large displacements and large rotations.
Updated Lagrangian For large displacements and large rotations.
Eulerian For large displacement, large rotations and moderately large strains.
Co-rotational For large displacements and large rotations.

Integration Schemes

- Stiffness** Default. 1-point (TPM3), 3-point (TPM6), 2x2 (QPM4, QPM8)
 Fine (see *Options*). 3x3 (QPM8), 3-point (TPM3).
Mass Default. 1-point (TPM3), 3-point (TPM6), 2x2 (QPM4, QPM8)
 Fine (see *Options*). 3x3 (QPM8), 3-point (TPM3).

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes fine integration rule.

- 34 Output element stress resultants.
- 36 Follower loads (see Notes)
- 54 Updated Lagrangian geometric nonlinearity.
- 55 Output strains as well as stresses.
- 87 Total Lagrangian geometric nonlinearity.
- 91 Invokes fine integration rule for mass matrix.
- 105 Lumped mass matrix.
- 123 Clockwise node numbering.
- 139 Output yielded Gauss points only
- 167 Eulerian geometric nonlinearity.
- 229 Co-rotational geometric nonlinearity

Notes on Use

1. The element formulations are based on the standard isoparametric approach. The variation of stresses within an element can be regarded as constant for the lower order (corner node only) elements, and linear for the higher (mid-side node) elements.
2. All elements pass the [patch test](#).
3. Non-conservative loading is available with these elements when using either Updated Lagrangian or Eulerian geometric nonlinear formulations together with the face loading (FLD).
4. Option 123 will not operate on a mesh with a mixture of clockwise and anti-clockwise elements, it is only applicable if **every** element is numbered **clockwise**. Surface normals should be visualised and if necessary corrected in the pre-processing stage.

Restrictions

- ☐ [Ensure mid-side node centrality](#)
- ☐ [Avoid excessive element curvature](#)
- ☐ Avoid excessive aspect ratio

Recommendations on Use

- The 8-noded element with a 2*2 Gauss rule is usually the most effective element, as the under-integration of the stiffness matrix prevents locking, which may occur either when the element is subjected to [parasitic shear](#), or as the material reaches the incompressible limit (elasto-plasticity). The Gauss point stresses are also sampled at the most accurate locations for the element. However, the element does possess one spurious zero energy mode. This mode is very rarely activated in linear analysis, but it may occur in both materially and geometrically nonlinear analyses. Therefore, a careful examination of the

solution should be performed, to check for spurious stress oscillations and peculiarities in the deformed configuration.

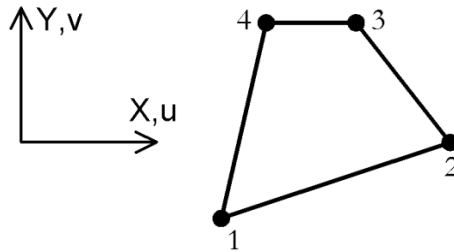
- The 8-noded element with a 3*3 Gauss rule may be used if a spurious mechanism is excited with the 2*2 Gauss rule.
- The 4-noded element should not be used for analyses where in-plane bending effects are significant as the element tends to lock in [parasitic shear](#) [C1], e.g. if QPM4 elements are employed to model a cantilever subject to a point load, the solution obtained will be over-stiff.

2D Plane Stress Continuum Element with Enhanced Strains

General

Element Name

QPM4M



Element Group

2D Continuum

Element

Plane Stress Continuum

Subgroup

Element

Description

A 2D isoparametric element with an assumed strain field. This mixed assumed strain element demonstrates a superior performance to QPM4 (see Notes). The elements are numerically integrated.

Number Of

Nodes

4, numbered anticlockwise.

Freedom

U, V: at each node.

Node

X, Y: at each node.

Coordinates

Geometric Properties

$t_1 \dots t_n$ Thickness at each node.

Material Properties

Linear

Isotropic:

MATERIAL PROPERTIES (Elastic: Isotropic)

Orthotropic:

MATERIAL PROPERTIES ORTHOTROPIC
(Elastic: Orthotropic Plane Stress)

Anisotropic:

MATERIAL PROPERTIES ANISOTROPIC 3
(Elastic: Anisotropic Thin Plate)

Rigidities:

RIGIDITIES 3 (Rigidities: Membrane/Thin Plate)

Matrix

Not applicable

Joint

Not applicable

Concrete

MATERIAL PROPERTIES NONLINEAR 94

		(Elastic: Isotropic, Plastic: Multi-Crack Concrete)
		MATERIAL PROPERTIES NONLINEAR 102
		(Elastic: Isotropic, Plastic: Smoothed Multi-Crack Concrete)
Elasto-Plastic	Stress resultant:	Not applicable
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Volumetric Crushing:	Not applicable
	Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
Creep		CREEP PROPERTIES (Creep)
Creep	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE) (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic Shrinkage	Not applicable	
		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Ogden:	MATERIAL PROPERTIES RUBBER OGDEN (Rubber: Ogden) (Rubber: Ogden)
	Mooney-Rivlin:	MATERIAL PROPERTIES RUBBER MOONEY_RIVLIN (Rubber: Mooney-Rivlin)
	Neo-Hookean:	MATERIAL PROPERTIES RUBBER

	Hencky:	NEO_HOOKEAN (Rubber: Neo-Hookean) MATERIAL PROPERTIES RUBBER HENCKY (Rubber: Hencky)
Generic Polymer	Isotropic	MATERIAL PROPERTIES NONLINEAR 89 (Generic Polymer Model)
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V: at nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py: at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	Face loads . Px, Py: local face axis pressures at nodes.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, $\Omega_x, \Omega_y, \Omega_z, \alpha_z$
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, ϕ_4 , Xcbf, Ycbf
Velocities	VELO	Velocities. Vx, Vy: at nodes.
Accelerations	ACCE	Accelerations. Ax, Ay: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. $\sigma_x, \sigma_y, \sigma_{xy}$: global stresses. $\epsilon_x, \epsilon_y, \gamma_{xy}$: global strains.
	SSIG	Initial stresses/strains at Gauss points. $\sigma_x, \sigma_y, \sigma_{xy}$: global stresses. $\epsilon_x, \epsilon_y, \gamma_{xy}$: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. $\sigma_x, \sigma_y, \sigma_{xy}$: global stresses.
	SSRG	Residual stresses at Gauss points. $\sigma_x, \sigma_y, \sigma_{xy}$: global stresses.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

Output

- Solver** Stress resultants: N_x , N_y , N_{xy} , N_{max} , N_{min} , β , N_s , N_e
 Stress (default): σ_x , σ_y , σ_{xy} , σ_{max} , σ_{min} , β , σ_s , σ_e (see [description of principal stresses](#))
 Strain: ϵ_x , ϵ_y , γ_{xy} , ϵ_{max} , ϵ_{min} , β , ϵ_s , ϵ_e
 Stretch (for rubber only): V_{11} , V_{22} , V_{12} , λ_1 , λ_2 , λ_3 , $\theta\lambda$, $\det F$. Where V_{ii} are components of the left stretch tensors, λ_i the principal stretches, $\theta\lambda$ the angle between the maximum principal stretch and the global X axis, and $\det F$ the determinant of the deformation gradient or volume ratio.
- Modeller** See [Results Tables \(Appendix K\)](#).

Local Axes

Not applicable (global axes are the reference).

Sign Convention

- ☐ [Standard 2D continuum element](#)

Formulation

Geometric Nonlinearity

- Total Lagrangian** For large displacements and large rotations.
- Updated Lagrangian** For large displacements and large rotations.
- Eulerian** For large displacements, large rotations and moderately large strains.
- Co-rotational** For large displacements and large rotations (large strains with rubber).

Integration Schemes

Stiffness Default.	2x2
Fine.	As default.
Mass Default.	2x2
Fine.	As default.

Mass Modelling

- ☐ Consistent mass (default).

- ☐ Lumped mass.

Options

- 34 Output element stress resultants.
- 36 Follower loads.
- 39 Stress smoothing for rubber material models.
- 54 Updated Lagrangian geometric nonlinearity.
- 55 Output strains as well as stresses.
- 87 Total Lagrangian geometric nonlinearity.
- 91 Invokes fine integration rule for mass matrix.
- 105 Lumped mass matrix.
- 123 Clockwise node numbering (see *Notes*).
- 139 Output yielded Gauss points only
- 167 Eulerian geometric nonlinearity.
- 225 Use alternative number of parameters for enhanced strain interpolation (see *Notes*).
- 229 Co-rotational geometric nonlinearity.

Notes on Use

1. The variation of stresses within an element can be regarded as linear.
2. The element passes the patch test and the large strain patch test for rubber.
3. The strain field for this element consists of two parts: the compatible strains derived from an assumed displacement field and the assumed enhanced strains (see *LUSAS Theory Manual*). The assumed enhanced strain field is defined using 5 or 4 parameters for linear and nonlinear applications respectively. Option 225 switches on the higher 5 parameter enhanced strain interpolation function for nonlinear analysis.
4. Non-conservative loading is available with these elements when using either Updated Lagrangian or Eulerian geometric nonlinear formulations together with the FLD loading facility. The load does not have to be normal to the face and may also vary over the face.
5. To apply a non-conservative (follower) pressure load (load type FLD) with co-rotational geometric nonlinearity, Option 36 must be specified. Note that this load must be normal to the face and constant for all the nodes of the element face.
6. The converged stresses for rubber are Kirchhoff stresses (see *LUSAS Theory Manual*).
7. When using the rubber material model, converged strain output is replaced by the left stretch tensor, the principal stretches and the angle defining these

principal directions. The value of $\det F = \lambda_1 \lambda_2$ (the Volume ratio) is only available for Gauss-point output. (Refer to the *LUSAS Theory Manual* for more details.)

8. For rubber, the iterative values of stress and strain are output in local co-rotated directions at the Gauss points only.
9. Option 123 will not operate on a mesh with a mixture of clockwise and anti-clockwise elements, it is only applicable if **every** element is numbered **clockwise**. Surface normals should be visualised and if necessary corrected in the pre-processing stage.
10. Using Option 123 with local loading types, such as FLD and UDL, will cause load reversal.
11. Convergence difficulties can sometimes arise when using enhanced strain elements with nonlinear materials, particularly if the material is elastic perfectly plastic or if a very shallow hardening curve is defined. In such cases it is recommended that the standard element formulation is used.
12. In analyses where significant in-plane bending is thermally induced it is recommended that a nonlinear solution is used. If a linear solution is required, then quadratic plane strain elements QPN8 are recommended.

Restrictions

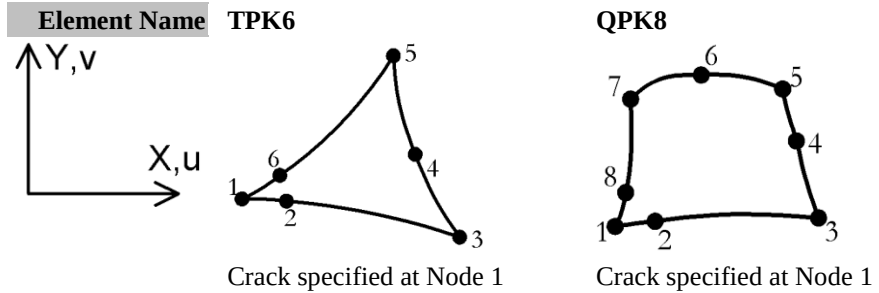
- ☐ [Avoid excessive aspect ratio](#)
- ☐ Rubber material models can only be applied in conjunction with the co-rotational formulation, Option 229.

Recommendations on Use

These elements exhibit an improved performance when compared with the parent element QPM4. The integration rules are the same as those given for QPM4, but the elements do not suffer from locking due to parasitic shear when the material approaches the incompressible limit. The elements are also free of any zero energy modes.

2D Plane Stress Continuum Crack Tip Elements

General



Element Group	2D Continuum
Element Subgroup	Plane Stress Continuum
Element Description	A family of 2D isoparametric crack tip elements where the crack tip can be located at any corner node. The mid-side nodes are moved to the quarter points to produce a singularity at the crack tip. The strains vary as the square root of $1/R$, where R is the distance from the crack tip. These elements are used at the crack tip only and should be mixed with the higher order plane strain continuum elements. The elements are numerically integrated.
Number Of Nodes	6 or 8 numbered anticlockwise.
End Releases	
Freedoms	U, V: at each node.
Node Coordinates	X, Y: at each node.

Geometric Properties

$t_1 \dots t_n$ Thickness at each node.

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic	MATERIAL PROPERTIES ORTHOTROPIC (Elastic: Orthotropic Plane Stress)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC 3 (Elastic: Anisotropic Thin Plate)
	Rigidities.	RIGIDITIES 3 (Rigidities: Membrane/Thin Plate)
Matrix	Not applicable	

Joint	Not applicable	
Concrete		MATERIAL PROPERTIES NONLINEAR 94 (Elastic: Isotropic, Plastic: Multi-Crack Concrete) MATERIAL PROPERTIES NONLINEAR 102 (Elastic: Isotropic, Plastic: Smoothed Multi-Crack Concrete)
Elasto-Plastic	Stress resultant:	Not applicable.
	Interface:	MATERIAL PROPERTIES NONLINEAR 27
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Volumetric Crushing:	Not applicable.
	Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
Creep		CREEP PROPERTIES (Creep)
	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE 2)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic Shrinkage	Not applicable	
		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable	
Generic Polymer	Isotropic	MATERIAL PROPERTIES NONLINEAR 89 (Generic Polymer Model)

Composite Not applicable

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V: at nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py: at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	Face loads . Px, Py: local face axis pressures at nodes.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Ω_x , Ω_y , Ω_z , α_z
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, ϕ_4 , Xcbf, Ycbf
Velocities	VELO	Velocities. Vx, Vy: at nodes.
Accelerations	ACCE	Accelerations. Ax, Ay: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. σ_x , σ_y , σ_{xy} : global stresses. ϵ_x , ϵ_y , γ_{xy} : global strains.
	SSIG	Initial stresses/strains at Gauss points. σ_x , σ_y , σ_{xy} : global stresses. ϵ_x , ϵ_y , γ_{xy} : global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σ_x , σ_y , σ_{xy} : global stresses.
	SSRG	Residual stresses at Gauss points. σ_x , σ_y , σ_{xy} : global stresses.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Stress resultants: Nx, Ny, Nxy, Nmax, Nmin, β , Ns, Ne
	Stress (default): σ_x , σ_y , σ_{xy} , σ_{max} , σ_{min} , β , σ_s , σ_e (see description of principal stresses)
	Strain: ϵ_x , ϵ_y , γ_{xy} , ϵ_{max} , ϵ_{min} , β , ϵ_s , ϵ_e

Modeller See [Results Tables \(Appendix K\)](#).

Local Axes

Not applicable (global axes are the reference).

Sign Convention

- ☐ [Standard 2D continuum element](#)

Formulation

Geometric Nonlinearity

- Total Lagrangian** For large displacements and large rotations.
- Updated Lagrangian** For large displacements and large rotations.
- Eulerian** For large displacements, large rotations and moderately large strains.
- Co-rotational** For large displacements and large rotations.

Integration Schemes

Stiffness	Default.	6-point (TPK6), 3x3 (QPK8)
	Fine (see <i>Options</i>).	12-point (TPK6).
Mass	Default.	6-point (TPK6), 3x3 (QPK8)
	Fine (see <i>Options</i>).	12-point (TPK6).

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes finer integration rule.
- 34** Output element stress resultants.
- 54** Updated Lagrangian geometric nonlinearity.
- 55** Output strains as well as stresses.
- 87** Total Lagrangian geometric nonlinearity.
- 91** Invokes fine integration rule for mass matrix.
- 105** Lumped mass matrix.
- 123** Clockwise node numbering.
- 139** Output yielded Gauss points only.

167 Eulerian geometric nonlinearity.

229 Co-rotational geometric nonlinearity.

Notes on Use

1. The element formulations are based on the standard isoparametric approach. Moving the mid-side nodes to the quarter points creates a singularity with theoretically infinite stress at the corner node.
2. Non-conservative loading is available with these elements when using either Updated Lagrangian or Eulerian geometric nonlinear formulations together with the FLD loading facility.
3. Option 123 will not operate on a mesh with a mixture of clockwise and anti-clockwise elements, it is only applicable if **every** element is numbered **clockwise**. Surface normals should be visualised and if necessary corrected in the pre-processing stage.
4. Using Option 123 with local loading types, such as FLD and UDL, will cause load reversal.

Restrictions

- ☐ [Avoid excessive element curvature](#)
- ☐ Avoid excessive aspect ratio

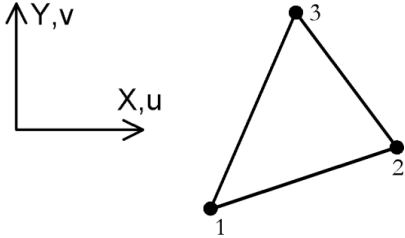
Recommendations on Use

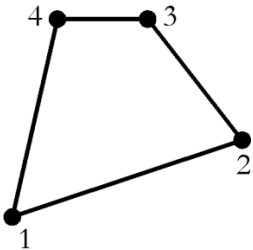
The QPK8 and TPK6 elements are specifically designed for application to fracture mechanics problems and may be used to model the singularities that occur at the crack tip. The mid-side nodes near the crack tip are shifted to the quarter point. This ensures a singularity is present at the crack tip and that the strains vary as $1/\sqrt{r}$ where r is the distance from the crack tip. The triangular TPK6 element is more effective than the quadrilateral element.

2D Plane Stress Explicit Dynamics Elements

General

Element Name	TPM3E	QPM4E





Element Group	2D Continuum
Element Subgroup	Plane Stress Continuum
Element Description	A family of 2D isoparametric elements for explicit dynamic analyses. The elements are numerically integrated.
Number Of Nodes	3 or 4 numbered anticlockwise.
End Releases	
Freedom Node	U, V: at each node. X, Y: at each node.
Coordinates	

Geometric Properties

$t_1 \dots t_n$ Thickness at each node.

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC (Elastic: Orthotropic Plane Stress)
	Anisotropic:	Not applicable
	Rigidities.	Not applicable
Matrix	Not applicable	
Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic	Stress resultant:	Not applicable
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61

		(Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Volumetric Crushing:	Not applicable
	Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
Creep	Not applicable	
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage	Not applicable	
Rubber	Not applicable	
Generic Polymer	Not applicable	
Composite	Not applicable	
Loading		
Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V: at each node.
Concentrated Loads	CL	Concentrated loads. Px, Py: at each node.
Element Loads	Not applicable.	
Distributed Loads	UDL FLD	Not applicable. Face loads . Px, Py: local face axis pressures at nodes.
Body Forces	CBF BFP, BFPE	Constant body forces for element. Xcbf, Ycbf, Ω_x , Ω_y , Ω_z , α_z Body force potentials at nodes/for element. 0, 0, 0, ϕ_4 , Xcbf, Ycbf
Velocities	VELO	Velocities. Vx, Vy: at nodes.
Accelerations	ACCE	Accelerations. Ax, Ay: at nodes.
Initial	SSI, SSIE	Initial stresses/strains at nodes/for element. σ_x ,

Stress/Strains		σ_y, σ_{xy} : global stresses. $\epsilon_x, \epsilon_y, \gamma_{xy}$: global strains.
	SSIG	Initial stresses/strains at Gauss points $\sigma_x, \sigma_y, \sigma_{xy}$: global stresses. $\epsilon_x, \epsilon_y, \gamma_{xy}$: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. $\sigma_x, \sigma_y, \sigma_{xy}$: global stresses.
	SSRG	Residual stresses at Gauss points. $\sigma_x, \sigma_y, \sigma_{xy}$: global stresses.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Stress (default): $\sigma_x, \sigma_y, \sigma_{xy}, \sigma_{\max}, \sigma_{\min}, \beta, \sigma_s, \sigma_e$ (see description of principal stresses)
	Strain: $\epsilon_x, \epsilon_y, \gamma_{xy}, \epsilon_{\max}, \epsilon_{\min}, \beta, \epsilon_s, \epsilon_e$
Modeller	See Results Tables (Appendix K) .

Local Axes

Not applicable (global axes are the reference).

Sign Convention

- ☐ [Standard 2D continuum element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	Not applicable.
Updated Lagrangian	Not applicable.
Eulerian	For large displacements, large rotations and moderately large strains.
Co-rotational	For large displacements and large rotations.

Integration Schemes

Stiffness	Default.	1-point (see <i>Notes</i>).
	Fine.	As default.
Mass	Default.	1-point (see <i>Notes</i>).
	Fine.	As default.

Mass Modelling

Lumped mass only (see *Notes*).

Options

- 34** Output element stress resultants.
- 55** Output strains as well as stresses.
- 105** Lumped mass matrix (see *Notes*).
- 139** Output yielded Gauss points only.

Notes on Use

1. The element formulations are based on the standard isoparametric approach. The variation of stresses within an element can be regarded as constant.
2. The system parameter HGVISC is used to restrict element mechanisms due to under-integration. The default value is usually sufficient.
3. The bulk viscosity coefficients are used to restrict numerical oscillations due to the traversal of stress waves. The default bulk viscosity coefficients (BULKLF and BULKQF) may be altered as SYSTEM parameters.
4. These elements **must** be used with the dynamic central difference scheme and a lumped mass matrix.
5. These elements are **not** applicable. for static or eigenvalue analyses.
6. Automatic time step calculations are implemented.
7. As the element geometry is always updated in an explicit dynamic analysis, a nonlinear solution is obtained. When using explicit dynamics elements nonlinear control must be specified.
8. If creep properties are defined, explicit time integration must be specified.
9. Non-conservative loading is invoked when the FLD loading facility is applied.
10. Rayleigh damping coefficients are not supported by these elements.
11. Constraint equations are not available for use with these elements.

12. Nodes **must** be specified in an anticlockwise order. Option 123 is **not** applicable for this element. When using Modeller ensure surface normal is in the +ve z direction.

Restrictions

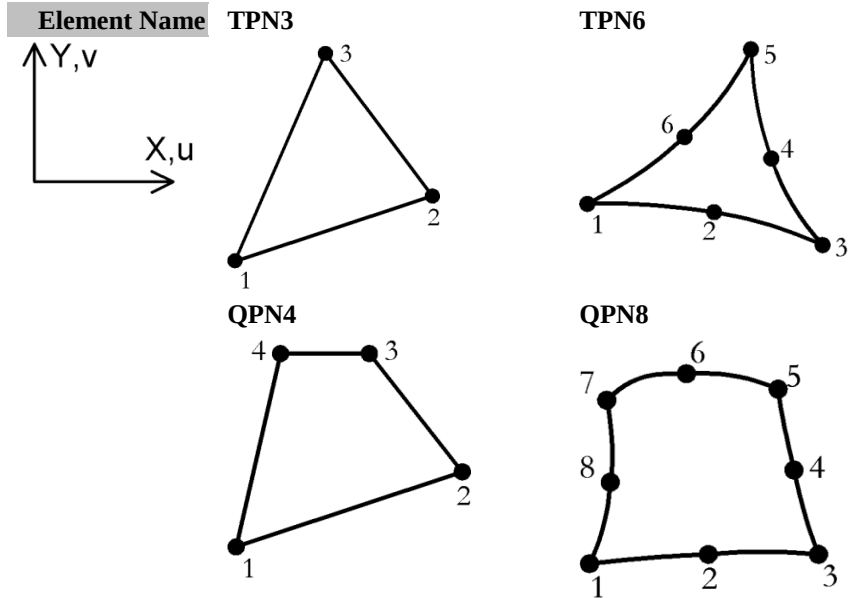
- ☐ [Avoid excessive aspect ratio](#)

Recommendations on Use

Explicit dynamics elements may be used to define surface boundaries which will be active in a slideline analysis.

2D Plane Strain Continuum Elements

General



Element Group	2D Continuum
Element Subgroup	Plane Strain Continuum
Element Description	A family of 2D isoparametric elements with higher order models capable of modelling curved boundaries. The elements are numerically integrated.
Number Of Nodes	3, 4, 6, or 8 numbered anticlockwise.
Freedom	U, V: at each node.
Node Coordinates	X, Y: at each node.

Geometric Properties

Not applicable (a unit thickness is assumed).

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC

		PLANE STRAIN (Elastic: Orthotropic Plane Strain)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC 4 (Not supported in LUSAS Modeller)
	Rigidities.	RIGIDITIES 4 (Not supported in LUSAS Modeller)
Matrix	Not applicable	
Joint	Not applicable	
Concrete		MATERIAL PROPERTIES NONLINEAR 94 (Elastic: Isotropic, Plastic: Multi Crack Concrete)
		MATERIAL PROPERTIES NONLINEAR 102 (Elastic: Isotropic, Plastic: Smoothed Multi-Crack Concrete)
Elasto-Plastic	Stress resultant:	Not applicable.
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Modified Mohr-Coulomb:	MATERIAL PROPERTIES MODIFIED MOHR_COULOMB (Elastic: Isotropic, Plastic: Mohr-Coulomb/Tresca, non-associative Hardening with tension/compression cut-off)
	Modified Cam-clay	MATERIAL PROPERTIES CAM_CLAY MODIFIED (Elastic: Isotropic, Plastic)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	MATERIAL PROPERTIES NONLINEAR 81 (Volumetric Crushing or Crushable Foam)
	Interface:	MATERIAL PROPERTIES NONLINEAR 27
	Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
Creep	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP

		(Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic		VISCO ELASTIC PROPERTIES
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable	
Generic	Isotropic	MATERIAL PROPERTIES NONLINEAR 89 (Generic Polymer Model)
Polymer		
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V: at nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py: at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	<u>Face Loads</u> . Px, Py: local face axis pressures at nodes.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, 0, 0, Ω_z , α_z
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, ϕ_4 , Xcbf, Ycbf
Velocities	VELO	Velocities. Vx, Vy: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. σ_x , σ_y , σ_{xy} , σ_z : global stresses. ϵ_x , ϵ_y , γ_{xy} : global strains.
	SSIG	Initial stresses/strains at Gauss points. σ_x , σ_y , σ_{xy} , σ_z : global stresses. ϵ_x , ϵ_y , γ_{xy} : global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σ_x , σ_y ,

	SSRG	σ_{xy} , σ_z : global stresses. Residual stresses at Gauss points. σ_x , σ_y , σ_{xy} , σ_z global stresses.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Stress (default): σ_x , σ_y , σ_{xy} , σ_z , σ_{max} , σ_{min} , β , σ_s , σ_e (see description of principal stresses) Strain: ϵ_x , ϵ_y , γ_{xy} , $\epsilon_z = 0$, ϵ_{max} , ϵ_{min} , β , ϵ_s , ϵ_e	
Modeller	See Results Tables (Appendix K) .	

Local Axes

Not applicable (global axes are the reference).

Sign Convention

- ☐ [Standard 2D continuum element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	For large displacements and large rotations.
Updated Lagrangian	For large displacements and large rotations.
Eulerian	For large displacements, large rotations and moderately large strains.
Co-rotational	For large displacements and large rotations.

Integration Schemes

Stiffness	Default.	1-point (TPN3), 3-point (TPN6), 2x2 (QPN4, QPN8)
	Fine (see <i>Options</i>).	3x3 (QPN8), 3-point (TPN3).
Mass	Default.	1-point (TPN3), 3-point (TPN6), 2x2 (QPN4, QPN8)
	Fine (see <i>Options</i>).	3x3 (QPN8), 3-point (TPN3).

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes finer integration rule.
- 36** Follower loads.
- 54** Updated Lagrangian geometric nonlinearity
- 55** Output strains as well as stresses.
- 87** Total Lagrangian geometric nonlinearity.
- 91** Invokes fine integration rule for mass matrix.
- 105** Lumped mass matrix.
- 123** Clockwise node numbering.
- 139** Output yielded Gauss points only.
- 167** Eulerian geometric nonlinearity.
- 229** Co-rotational geometric nonlinearity

Notes on Use

1. The element formulations are based on the standard isoparametric approach. The variation of stresses within an element can be regarded as constant for the lower order (corner node only) elements, and linear for the higher order (mid-side node) elements.
2. All elements pass the [patch test](#).
3. Non-conservative loading is available with these elements when using either Updated Lagrangian or Eulerian geometric nonlinear formulations together with the FLD loading facility.
4. Option 123 will not operate on a mesh with a mixture of clockwise and anti-clockwise elements, it is only applicable if **every** element is numbered **clockwise**. Surface normals should be visualised and if necessary corrected in the pre-processing stage.
5. Using Option 123 with local loading types, such as FLD and UDL, will cause load reversal.

Restrictions

- ☐ [Ensure mid-side node centrality](#)
- ☐ [Avoid excessive element curvature](#)
- ☐ Avoid excessive aspect ratio

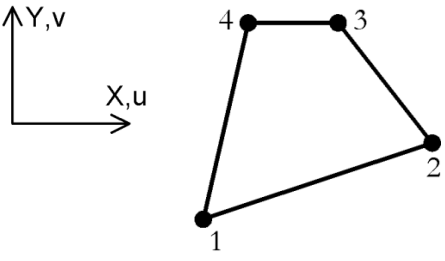
Recommendations on Use

- The 8-noded element with a 2*2 Gauss rule is usually the most effective element, as the under-integration of the stiffness matrix prevents locking, which may occur either when the element is subjected to [parasitic shear](#), or as the material reaches the incompressible limit (elasto-plasticity). The Gauss point stresses are also sampled at the most accurate locations for the element. However, the element does possess one spurious zero energy mode. This mode is very rarely activated in linear analysis, but it may occur in both materially and geometrically nonlinear analyses. Therefore, a careful examination of the solution should be performed, to check for spurious stress oscillations and peculiarities in the deformed configuration.
- The 8-noded element with a 3*3 Gauss rule may be used if a spurious mechanism is excited with the 2*2 Gauss rule.
- The 4-noded element should not be used for analyses where in-plane bending effects are significant as the element tends to lock in [parasitic shear](#), e.g. if QPN4 elements are employed to model a cantilever subject to a point load, the solution obtained will be over-stiff.

2D Plane Strain Continuum Element with Enhanced Strains

General

Element Name	QPN4M



Element Group	2D Continuum
Element Subgroup	Plane Strain Continuum
Element Description	A 2D isoparametric element with an assumed strain field. This mixed assumed strain element demonstrates a superior performance to QPN4 (see Notes). The element is numerically integrated.
Number Of Nodes	4, numbered anticlockwise.
Freedom	U, V: at each node.
Node Coordinates	X, Y: at each node.

Geometric Properties

Not applicable (a unit thickness is assumed).

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC PLANE STRAIN (Elastic: Orthotropic Plane Strain)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC 4 (Not supported in LUSAS Modeller)
	Rigidities.	RIGIDITIES 4 (Not supported in LUSAS Modeller)
Matrix	Not applicable	
Joint	Not applicable	

Concrete		MATERIAL PROPERTIES NONLINEAR 94 (Elastic: Isotropic, Plastic: Multi-Crack Concrete)
		MATERIAL PROPERTIES NONLINEAR 102 (Elastic: Isotropic, Plastic: Smoothed Multi-Crack Concrete)
Elasto-Plastic	Stress resultant:	Not applicable.
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Modified Mohr-Coulomb:	MATERIAL PROPERTIES MODIFIED MOHR_COULOMB (Elastic: Isotropic, Plastic: Mohr-Coulomb/Tresca, non-associative Hardening with tension/compression cut-off)
	Modified Cam-clay	MATERIAL PROPERTIES CAM_CLAY MODIFIED (Elastic: Isotropic, Plastic)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	MATERIAL PROPERTIES NONLINEAR 81 (Volumetric Crushing or Crushable Foam)
	Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
	Creep	CREEP PROPERTIES (Creep)
	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
Damage	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
		DAMAGE PROPERTIES SIMO, OLIVER (Damage)

Viscoelastic		VISCO ELASTIC PROPERTIES
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Ogden	MATERIAL PROPERTIES RUBBER OGDEN (Rubber: Ogden)
	Mooney-Rivlin	MATERIAL PROPERTIES RUBBER MOONEY_RIVLIN (Rubber: Mooney-Rivlin)
	Neo-Hookean	MATERIAL PROPERTIES RUBBER NEO_HOOKEAN (Rubber: Neo-Hookean)
	Hencky	MATERIAL PROPERTIES RUBBER HENCKY (Rubber: Hencky)
Generic Polymer	Isotropic	MATERIAL PROPERTIES NONLINEAR 89 (Generic Polymer Model)
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V: at nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py: at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	Face loads . Px, Py: local face axis pressures at nodes.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, 0, 0, Ω_z , α_z
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, φ_4 , Xcbf, Ycbf
Velocities	VELO	Velocities. Vx, Vy: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. σ_x , σ_y , σ_{xy} , σ_z : global stresses. ϵ_x , ϵ_y , γ_{xy} : global strains.
	SSIG	Initial stresses/strains at Gauss points. σ_x , σ_y , σ_{xy} , σ_z : global stresses. ϵ_x , ϵ_y , γ_{xy} : global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σ_x , σ_y , σ_{xy} , σ_z : global stresses.
	SSRG	Residual stresses at Gauss points. σ_x , σ_y , σ_{xy} , σ_z global stresses.

Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Stress (default): $\sigma_x, \sigma_y, \sigma_{xy}, \sigma_z, \sigma_{\max}, \sigma_{\min}, \beta, \sigma_s, \sigma_e$ (see description of principal stresses) Strain: $\epsilon_x, \epsilon_y, \gamma_{xy}, \epsilon_z = 0, \epsilon_{\max}, \epsilon_{\min}, \beta, \epsilon_s, \epsilon_e$ Stretch (for rubber only): $V_{11}, V_{22}, V_{12}, \lambda_1, \lambda_2, \lambda_3=1, \theta\lambda, \det F$. Where V_{ii} are components of the left stretch tensors, λ_i the principal stretches, $\theta\lambda$ the angle between the maximum principal stretch and the global X axis, and $\det F$ the determinant of the deformation gradient or volume ratio.
Modeller	See Results Tables (Appendix K) .

Local Axes

Not applicable (global axes are the reference).

Sign Convention

- ☐ [Standard 2D continuum element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	For large displacements and large rotations.
Updated Lagrangian	For large displacements and large rotations.
Eulerian	For large displacements, large rotations and moderately large strains.
Co-rotational	For large displacements and large rotations (large strains with rubber).

Integration Schemes

Stiffness	Default. 2x2
	Fine. As default.
Mass	Default. 2x2

Fine. As default.

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Output

- 36 Follower loads.
- 39 Stress smoothing for rubber material models.
- 54 Updated Lagrangian geometric nonlinearity.
- 55 Output strains as well as stresses.
- 87 Total Lagrangian geometric nonlinearity.
- 91 Invokes fine integration rule for mass matrix.
- 105 Lumped mass matrix.
- 123 Clockwise node numbering.
- 139 Output yielded Gauss points only
- 167 Eulerian geometric nonlinearity.
- 225 Use alternative number of parameters for enhanced strain interpolation (see *Notes*).
- 229 Co-rotational geometric nonlinearity.

Notes on Use

1. The element formulation is based on the standard
2. The element passes the patch test and the large strain patch test for rubber.
3. The strain field for this element consists of two parts: the compatible strains derived from an assumed displacement field and the assumed enhanced strains; see *LUSAS Theory Manual*. The assumed enhanced strain field is defined using 5 or 4 parameters for linear and nonlinear applications respectively. Option 225 switches on the higher 5 parameter enhanced strain interpolation function for nonlinear analysis.
4. Non-conservative loading is available with these elements when using either Updated Lagrangian or Eulerian geometric nonlinear formulations together with the FLD loading facility. The load does not have to be normal to the face and may also vary over the face.
5. To apply a non-conservative (follower) pressure load (load type FLD) with co-rotational geometric nonlinearity, Option 36 must be specified. Note that this load should be normal to the face and constant for all the nodes of the element face.

6. The converged stresses for rubber are [Kirchhoff](#) stresses (see *LUSAS Theory Manual*).
7. Option 39 is used to smooth the stress output. It is particularly useful when the rubber material model is applied and the element is under very high compression where oscillatory stresses may appear (checker-board pattern).
8. When using the rubber material model, converged strain output is replaced by the left stretch tensor, the principal stretches and the angle defining these principal directions. The value of $\det F = \lambda_1 \lambda_2$ (the Volume ratio) is only available for Gauss-point output. (Refer to the *LUSAS Theory Manual* for more details.)
9. For rubber, the iterative values of stress and strain are output in local co-rotated directions at the Gauss points only.
10. Option 123 will not operate on a mesh with a mixture of clockwise and anti-clockwise elements, it is only applicable if **every** element is numbered **clockwise**. Surface normals should be visualised and if necessary corrected in the pre-processing stage.
11. Using Option 123 with local loading types, such as FLD and UDL, will cause load reversal.
12. Convergence difficulties can sometimes arise when using enhanced strain elements with nonlinear materials, particularly if the material is elastic-perfectly plastic or if a very shallow hardening curve is defined. In such cases it is recommended that the standard element formulation is used.
13. In analyses where significant in-plane bending is thermally induced it is recommended that a nonlinear solution is used. If a linear solution is required, then quadratic plane strain elements QPN8 are recommended.

Restrictions

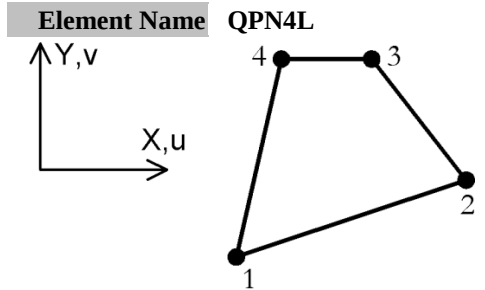
- ☐ Rubber material models can only be applied in conjunction with the co-rotational formulation, Option 229.
- ☐ Avoid excessive aspect ratio

Recommendations on Use

These elements exhibit an improved performance when compared with the parent element QPN4. The integration rules are the same as the parent element. The elements do not suffer from locking due to parasitic shear or when the material approaches the incompressible limit. The elements are also free of any zero energy modes.

2D Plane Strain Continuum Element for Large Strains

General



Element Group	2D Continuum
Element Subgroup	Plane Strain Continuum
Element Description	A 2D isoparametric element incorporating an internal pressure variable. This element should be used for analyses involving large strains. The element is numerically integrated 4, numbered anticlockwise.
Number Of Nodes	4, numbered anticlockwise.
Freedom	U, V: at each node.
Node Coordinates	X, Y: at each node.

Geometric Properties

Not applicable (a unit thickness is assumed).

Material Properties

Linear	Not applicable
Matrix	Not applicable
Joint	Not applicable
Concrete	Not applicable
Elasto-Plastic	Implicit
	Optimised Von Mises
	Stress Potential
	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic)
	STRESS POTENTIAL VON_MISES (Isotropic: von Mises)
Creep	Not applicable
Damage	Not applicable

Viscoelastic	Not applicable	
Shrinkage	Not applicable	
Rubber	Ogden	MATERIAL PROPERTIES RUBBER OGDEN (Rubber: Ogden)
	Mooney-Rivlin	MATERIAL PROPERTIES RUBBER MOONEY_RIVLIN (Rubber: Mooney-Rivlin)
	Neo-Hookean	MATERIAL PROPERTIES RUBBER NEO_HOOKEAN (Rubber: Neo-Hookean)
	Hencky	MATERIAL PROPERTIES RUBBER HENCKY (Rubber: Hencky)
Generic Polymer	Not applicable	
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V: at nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py: at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	Face loads . Px, Py: local face axis pressures at nodes.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, 0,0, Ω_z , α_z
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, ϕ_4 , Xcbf, Ycbf
Velocities	VELO	Velocities. Vx, Vy: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. σ_x , σ_y , σ_{xy} , σ_z : global stresses. ϵ_x , ϵ_y , γ_{xy} : global strains.
	SSIG	Initial stresses/strains at Gauss points. σ_x , σ_y , σ_{xy} , σ_z : global stresses. ϵ_x , ϵ_y , γ_{xy} : global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σ_x , σ_y , σ_{xy} , σ_z : global stresses.
	SSRG	Residual stresses at Gauss points. σ_x , σ_y , σ_{xy} , σ_z global stresses.

Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Stress (default): σ_x , σ_y , σ_{xy} , σ_z , σ_{max} , σ_{min} , β , σ_s , σ_e (see description of principal stresses) Principal stretches, λ_1 , λ_2 , $\lambda_3=1$, $\theta\lambda$, $\det F$. Where V_{ii} are components of the left stretch tensors, λ_i the principal stretches, $\theta\lambda$ the angle between the maximum principal stretch and the global X axis, and $\det F$ the determinant of the deformation gradient or volume ratio.
Modeller	See Results Tables (Appendix K) .

Local Axes

Not applicable (global axes are the reference).

Sign Convention

- ☐ [Standard 2D continuum element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	Not applicable.
Updated Lagrangian	Not applicable.
Eulerian	For large displacements and large strains.
Co-rotational	For large displacements and large rotations.

Integration Schemes

Stiffness	Default.	2x2
	Fine.	As default.
Mass	Default.	2x2
	Fine.	As default.

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 55 Output stretches as well as stresses.
- 91 Invokes fine integration rule for mass matrix.
- 105 Lumped mass matrix.
- 123 Clockwise node numbering.

Notes on Use

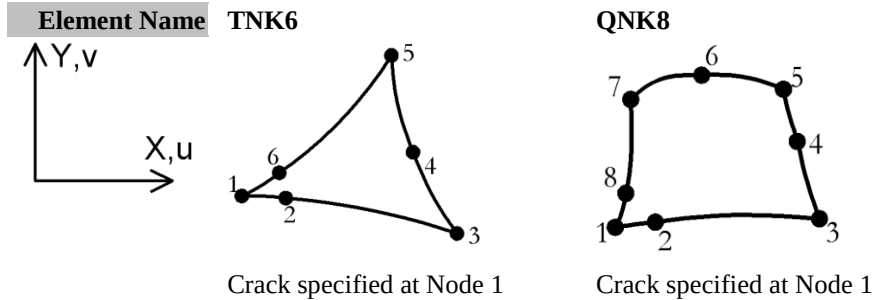
1. The element formulations are based on the standard [isoparametric](#) approach. The variation of stresses within an element can be regarded as linear.
2. The element passes the large strain patch test for rubber.
3. Non-conservative loading is available with this element when using FLD loading.
4. The stresses output are [Kirchhoff](#) stresses (see *LUSAS Theory Manual*).
5. Stretch output consists of the principal stretches and the angle defining the principal directions. The value of $\det F = \lambda_1 \lambda_2$ is also output. (Refer to the *LUSAS Theory Manual*.)
6. Option 123 will not operate on a mesh with a mixture of clockwise and anti-clockwise elements, it is only applicable if **every** element is numbered **clockwise**. Surface normals should be visualised and if necessary corrected in the pre-processing stage.
7. Using Option 123 with local loading types, such as FLD and UDL, will cause load reversal.
8. This element is based on a formulation that tackles the problem of volumetric locking in a different way to that used in QPN4M. It should be preferred to the QPN4M in cases where Eulerian description (with a current configuration taken as reference) is more appropriate than the co-rotational description (e.g. inflation problems).

Restrictions

- ☐ [Avoid excessive aspect ratio](#)
- ☐ Avoid non-uniform initial and thermal strains with coarse meshes.

2D Plane Strain Continuum Crack Tip Elements

General



Element Group	2D Continuum
Element Subgroup	Plane Strain Continuum
Element Description	A family of 2D isoparametric crack tip elements where the crack tip can be located at any corner node. The mid-side nodes are moved to the quarter points to produce a singularity at the crack tip. The strains vary as the square root of $1/R$, where R is the distance from the crack tip. These elements are used at the crack tip only and should be mixed with the higher order plane strain continuum elements. The elements are numerically integrated.
Number Of Nodes	6 or 8, numbered anticlockwise.
Freedoms	U, V: at each node.
Node Coordinates	X, Y: at each node.

Geometric Properties

Not applicable (a unit thickness is assumed).

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC PLANE STRAIN (Elastic: Orthotropic Plane Strain)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC 4 (Not supported in LUSAS Modeller)
	Rigidities.	RIGIDITIES 4 (Not supported in LUSAS Modeller)

Matrix Not applicable

Joint Not applicable

Concrete		MATERIAL PROPERTIES NONLINEAR 94 (Elastic: Isotropic, Plastic: Multi-Crack Concrete)
		MATERIAL PROPERTIES NONLINEAR 102 (Elastic: Isotropic, Plastic: Smoothed Multi-Crack Concrete)
Elasto-Plastic	Stress resultant:	Not applicable.
	Interface:	MATERIAL PROPERTIES NONLINEAR 27
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Modified Mohr-Coulomb:	MATERIAL PROPERTIES MODIFIED MOHR_COULOMB (Elastic: Isotropic, Plastic: Mohr-Coulomb/Tresca, non-associative Hardening with tension/compression cut-off)
	Modified Cam-clay	MATERIAL PROPERTIES CAM_CLAY MODIFIED (Elastic: Isotropic, Plastic)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	MATERIAL PROPERTIES NONLINEAR 81 (Volumetric Crushing or Crushable Foam)
	Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
Creep		CREEP PROPERTIES (Creep)

	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic		VISCO ELASTIC PROPERTIES
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable	
Generic Polymer	Isotropic	MATERIAL PROPERTIES NONLINEAR 89 (Generic Polymer Model)
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V: at nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py: at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	Face loads . Px, Py: local face axis pressures at nodes.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, 0, 0, Ω_z , α_z
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, φ_4 , Xcbf, Ycbf
Velocities	VELO	Velocities. Vx, Vy: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. σ_x , σ_y , σ_{xy} , σ_z : global stresses. ϵ_x , ϵ_y , γ_{xy} : global strains.

	SSIG	Initial stresses/strains at Gauss points. σ_x , σ_y , σ_{xy} , σ_z : global stresses. ϵ_x , ϵ_y , γ_{xy} : global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σ_x , σ_y , σ_{xy} , σ_z : global stresses.
	SSRG	Residual stresses at Gauss points. σ_x , σ_y , σ_{xy} , σ_z : global stresses.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, T ₀ , 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Stress (default): σ_x , σ_y , σ_{xy} , $\sigma_{\max}$, $\sigma_{\min}$, β , σ_s , σ_e (see description of principal stresses)
	Strain: ϵ_x , ϵ_y , γ_{xy} , $\epsilon_{\max}$, $\epsilon_{\min}$, β , ϵ_s , ϵ_e
Modeller	See Results Tables (Appendix K) .

Local Axes

Not applicable (global axes are the reference).

Sign Convention

- ☐ [Standard 2D continuum element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	For large displacements and large rotations.
Updated Lagrangian	For large displacements and large rotations.
Eulerian	For large displacements, large rotations and moderately large strains.
Co-rotational	For large displacements and large rotations.

Integration Schemes

Stiffness	Default.	6-point (TNK6), 3x3 (QNK8)

Fine (see *Options*). 12-point (TNK6)
Mass Default. 6-point (TNK6), 3x3 (QNK8)
Fine (see *Options*). 12-point (TNK6)

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes finer integration rule.
- 54** Updated Lagrangian geometric nonlinearity.
- 55** Output strains as well as stresses.
- 87** Total Lagrangian geometric nonlinearity.
- 91** Invokes fine integration rule for mass matrix.
- 105** Lumped mass matrix.
- 123** Clockwise node numbering.
- 139** Output yielded Gauss points only.
- 167** Eulerian geometric nonlinearity.
- 229** Co-rotational geometric nonlinearity.

Notes on Use

1. The element formulations are based on the standard isoparametric approach. Moving the mid-side nodes to the quarter points creates a singularity with theoretically infinite stress at the corner node.
2. Non-conservative loading is available with these elements when using either Updated Lagrangian or Eulerian geometric nonlinear formulations together with the FLD loading facility.
3. Option 123 will not operate on a mesh with a mixture of clockwise and anti-clockwise elements, it is only applicable if **every** element is numbered **clockwise**. Surface normals should be visualised and if necessary corrected in the pre-processing stage.
4. Using Option 123 with local loading types, such as FLD and UDL, will cause load reversal.

Restrictions

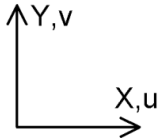
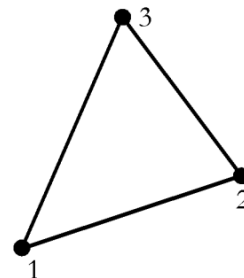
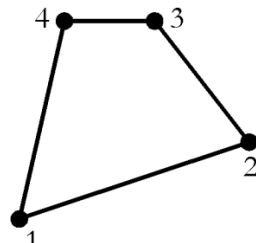
- ☐ [Avoid excessive element curvature](#)
- ☐ Avoid excessive aspect ratio

Recommendations on Use

Elements QNK8 and TNK6 are specifically designed for application to fracture mechanics problems and may be used to model the singularities that occur at the crack tip. The mid-side nodes near the crack tip are shifted to the quarter point. This ensures a singularity is present at the crack tip and that the strains vary as $1/\sqrt{r}$ where r is the distance from the crack tip. The triangular TNK6 element is more effective than the quadrilateral element.

2D Plane Strain Explicit Dynamics Elements

General

Element Name	TPN3E	QPN4E
		
Element Group	2D Continuum	
Element Subgroup	Plane Strain Continuum	
Element Description	A family of 2D isoparametric elements for explicit dynamic analyses. The elements are numerically integrated.	
Number Of Nodes	3 or 4 numbered anticlockwise.	
Freedom	U, V: at each node.	
Node Coordinates	X, Y: at each node.	

Geometric Properties

Not applicable (a unit thickness is assumed).

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC PLANE STRAIN (Elastic: Orthotropic Plane Strain)
	Anisotropic:	Not applicable.
	Rigidities.	Not applicable.
Matrix	Not applicable	
Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic	Stress resultant:	Not applicable.
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61

		(Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Modified Mohr-Coulomb:	MATERIAL PROPERTIES MODIFIED MOHR_COULOMB (Elastic: Isotropic, Plastic: Mohr-Coulomb/Tresca, non-associative Hardening with tension/compression cut-off)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	MATERIAL PROPERTIES NONLINEAR 81 (Volumetric Crushing or Crushable Foam)
	Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
Creep		CREEP PROPERTIES (Creep) (see <i>Notes</i>)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic		VISCO ELASTIC PROPERTIES
Shrinkage	Not applicable	
Rubber	Not applicable	
Generic Polymer	Not applicable	
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V: at each node.
Concentrated Loads	CL	Concentrated loads. Px, Py: at each node.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	Face loads . Px, Py: local face axis pressures at nodes.

Body Forces	CBF	Constant body forces for element. X_{cbf} , Y_{cbf} , 0 , 0 , Ω_z , α_z
	BFP, BFPE	Body force potentials at nodes/for element. 0 , 0 , ϕ_4 , X_{cbf} , Y_{cbf}
Velocities	VELO	Velocities. V_x , V_y : at nodes.
Accelerations	ACCE	Acceleration A_x , A_y : at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. σ_x , σ_y , σ_{xy} , σ_z : global stresses. ϵ_x , ϵ_y , γ_{xy} global strains.
	SSIG	Initial stresses/strains at Gauss points. σ_x , σ_y , σ_{xy} , σ_z : global stresses. ϵ_x , ϵ_y , γ_{xy} : global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σ_x , σ_y , σ_{xy} , σ_z : global stresses.
	SSRG	Residual stresses at Gauss points. σ_x , σ_y , σ_{xy} , σ_z : global stresses.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T , 0 , 0 , 0 , T_0 , 0 , 0 , 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Stress (default): σ_x , σ_y , σ_{xy} , σ_z , σ_{max} , σ_{min} , β , σ_s , σ_e (see description of principal stresses)
	Strain: ϵ_x , ϵ_y , γ_{xy} , ϵ_{max} , ϵ_{min} , β , ϵ_s , ϵ_e
Modeller	See Results Tables (Appendix K) .

Local Axes

Not applicable (global axes are the reference).

Sign Convention

- ☐ [Standard 2D continuum element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	Not applicable.
Updated Lagrangian	Not applicable.
Eulerian	For large displacements, large rotations and moderately large strains.
Co-rotational	For large displacements and large rotations.

Integration Schemes

Stiffness	Default.	1-point (see <i>Notes</i>).
	Fine.	As default.
Mass	Default.	1-point (see <i>Notes</i>).
	Fine.	As default.

Mass Modelling

- ☐ Lumped mass only (see *Notes*).

Options

- 55 Output strains as well as stresses.
- 105 Lumped mass matrix (see *Notes*).
- 139 Output yielded Gauss points only.

Notes on Use

1. The element formulations are based on the standard
2. The system parameter HGVISC is used to restrict element mechanisms due to under-integration. The default value is usually sufficient.
3. The bulk viscosity coefficients are used to restrict numerical oscillations due to the traversal of stress waves. The default bulk viscosity coefficients (BULKLF and BULKQF) may be altered as SYSTEM parameters.
4. These elements must be used with a dynamic central difference scheme and a lumped mass matrix in order to obtain the maximum efficiency from the numerical algorithms.
5. These elements are Not applicable. for static or eigenvalue analyses.
6. Automatic time step calculations are implemented.

7. As the element geometry is always updated in an explicit dynamic analysis, a nonlinear solution is obtained. When using explicit dynamics elements NONLINEAR CONTROL must be specified.
8. If CREEP PROPERTIES are defined, explicit time integration must be specified in VISCOUS CONTROL.
9. Non-conservative loading is invoked when the FLD loading facility is applied.
10. Rayleigh damping coefficients are not supported by these elements.
11. Constraint equations are not available for use with these elements.
12. Nodes **must** be specified in an anticlockwise order. Option 123 is **not** applicable for this element. When using Modeller ensure surface normal is in the +ve z direction.

Restrictions

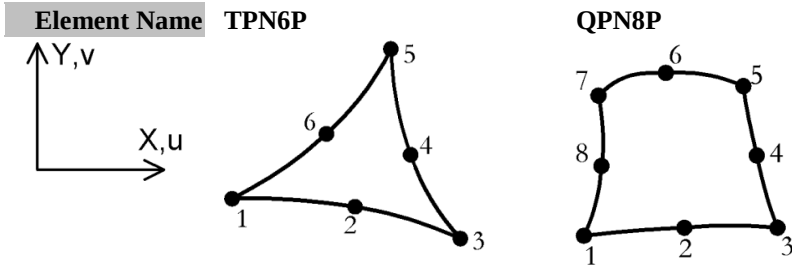
- ❑ [Avoid excessive aspect ratio](#)

Recommendations on Use

Explicit dynamics elements may be used to define surface boundaries which will be active in a slideline analysis.

2D Plane Strain Two Phase Continuum Elements

General



Element Group	2D Continuum
Element Subgroup	Plane Strain Continuum
Element Description	A family of 2D isoparametric elements with higher order models capable of modelling curved boundaries. The elements are numerically integrated.
Number Of Nodes	6 or 8 numbered anticlockwise.
Freedom	U, V, P at corner nodes. U, V at midside nodes.
Node Coordinates	X, Y: at each node.

Geometric Properties

Not applicable (a unit thickness is assumed).

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC PLANE STRAIN (Elastic: Orthotropic Plane Strain)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC 4 (Not supported in LUSAS Modeller)
	Rigidities.	RIGIDITIES 4 (Not supported in LUSAS Modeller)
Matrix	Not applicable	
Joint	Not applicable	
Concrete		MATERIAL PROPERTIES NONLINEAR 94 (Elastic: Isotropic, Plastic: Multi Crack Concrete)
		MATERIAL PROPERTIES NONLINEAR 102

		(Elastic: Isotropic, Plastic: Smoothed Multi Crack Concrete)
Elasto-Plastic	Stress resultant:	Not applicable.
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Modified Mohr-Coulomb:	MATERIAL PROPERTIES MODIFIED MOHR_COULOMB (Elastic: Isotropic, Plastic: Mohr-Coulomb/Tresca, non-associative Hardening with tension/compression cut-off)
	Modified Cam-clay	MATERIAL PROPERTIES CAM_CLAY MODIFIED (Elastic: Isotropic, Plastic)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	MATERIAL PROPERTIES NONLINEAR 81 (Volumetric Crushing or Crushable Foam)
	Interface	MATERIAL PROPERTIES NONLINEAR 27
	Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
Creep		CREEP PROPERTIES (Creep)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic		VISCOELASTIC PROPERTIES
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable	
Generic Polymer		MATERIAL PROPERTIES NONLINEAR 89 (Generic Polymer Model)
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, P at corner nodes. U, V at midside nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py, Q at corner nodes. Px, Py at midside nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	Face Loads . Px, Py, Q: face pressures/flux per unit area at corner nodes relative to local face axes. Px, Py: face pressures at midside nodes relative to local face axes.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, 0, 0, Ω_z , α_z , gx, gy (see Notes on Use)
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, ϕ_4 , Xcbf, Ycbf, gx, gy (see Notes on Use)
Velocities	VELO	Velocities. Vx, Vy: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. σ_x , σ_y , σ_{xy} , σ_z , σ_p global stresses. ϵ_x , ϵ_y , γ_{xy} : global strains.
	SSIG	Initial stresses/strains at Gauss points. σ_x , σ_y , σ_{xy} , σ_z , σ_p : global stresses. ϵ_x , ϵ_y , γ_{xy} : global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σ_x , σ_y , σ_{xy} , σ_z , σ_p : global stresses.
	SSRG	Residual stresses at Gauss points. σ_x , σ_y , σ_{xy} , σ_z , σ_p global stresses.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Stress (default): σ_x , σ_y , σ_{xy} , σ_z , σ_p , σ_{max} , σ_{min} , β , σ_s , σ_e (see description of principal stresses)

Strain: $\epsilon_x, \epsilon_y, \gamma_{xy}, \epsilon_z = 0, \epsilon_v, \epsilon_{\max}, \epsilon_{\min}, \beta, \epsilon_s, \epsilon_e$

Modeller See [Results Tables \(Appendix K\)](#).

Local Axes

Not applicable (global axes are the reference).

Sign Convention

- ☐ [Standard 2D continuum element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian For large displacements and large rotations.

Updated Lagrangian For large displacements and large rotations.

Eulerian For large displacements, large rotations and moderately large strains.

Co-rotational For large displacements and large rotations.

Integration Schemes

Stiffness Default. 3-point (TPN6P), 2x2 (QPN8P)
Fine (see *Options*). 3x3 (QPN8P)

Mass Default. 3-point (TPN6P), 2x2 (QPN8P)
Fine (see *Options*). 3x3 (QPN8P)

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes finer integration rule.
- 36** Follower loads.
- 54** Updated Lagrangian geometric nonlinearity
- 55** Output strains as well as stresses
- 87** Total Lagrangian geometric nonlinearity.

- 91 Invokes fine integration rule for mass matrix.
- 105 Lumped mass matrix.
- 123 Clockwise node numbering.
- 139 Output yielded Gauss points only.
- 167 Eulerian geometric nonlinearity.
- 229 Co-rotational geometric nonlinearity.

Notes on Use

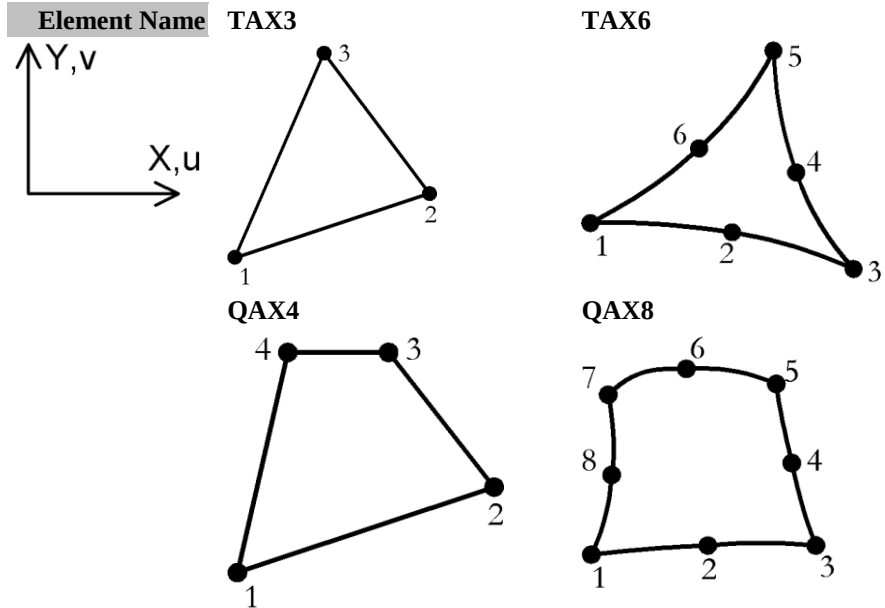
1. Two phase material parameters must be used with these elements for undrained and consolidation analysis.
2. The element formulations are based on the standard isoparametric approach. The variation of isoparametric stresses and pore pressures within an element can be considered linear.
3. All elements pass the [patch test](#).
4. Option 123 will not operate on a mesh with a mixture of clockwise and anti-clockwise elements, it is only applicable if **every** element is numbered **clockwise**. Surface normals should be visualised and if necessary corrected in the pre-processing stage. Using Option 123 with local loading types, such as FLD and UDL, will cause load reversal.
5. Non-conservative loading is available with these elements when using Updated Lagrangian, Eulerian or co-rotational (with OPTION 36) geometric nonlinear formulations together with the FLD loading facility.
6. The global components of gravity acting on the fluid phase are defined by gx and gy under CBF and BFP loading.

Restrictions

- ☐ [Ensure mid-side node centrality](#)
- ☐ [Avoid excessive element curvature](#)
- ☐ Avoid excessive aspect ratio

2D Axisymmetric Solid Continuum Elements

General



Element Group

2D Continuum

Element Subgroup

Axisymmetric Solid

Element Description

A family of 2D **isoparametric** elements with higher order models capable of modelling curved boundaries. The formulations apply over a unit radian segment of the structure and the loading and boundary conditions are axisymmetric. By default, the Y-axis is taken as the axis of symmetry. The elements are numerically integrated.

Number Of Nodes

3, 4, 6, or 8 numbered anticlockwise.

Freedom

U, V: at each node.

Node

X, Y: at each node.

Coordinates

Geometric Properties

Not applicable (a unit radian segment is assumed).

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC AXISYMMETRIC (Elastic: orthotropic Axisymmetric)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC 4 (Not supported in LUSAS Modeller)
	Rigidities.	Not applicable.
Matrix	Not applicable	
Joint	Not applicable	
Concrete		MATERIAL PROPERTIES NONLINEAR 94 (Elastic: Isotropic, Plastic: Multi-Crack Concrete)
		MATERIAL PROPERTIES NONLINEAR 102 (Elastic: Isotropic, Plastic: Smoothed Multi-Crack Concrete)
Elasto- Plastic	Stress resultant:	Not applicable.
	Interface:	MATERIAL PROPERTIES NONLINEAR 27.
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker- Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr- Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Modified Mohr- Coulomb:	MATERIAL PROPERTIES MODIFIED MOHR_COULOMB (Elastic: Isotropic, Plastic: Mohr-Coulomb/Tresca, non-associative Hardening with tension/compression cut-off)
	Modified Cam-clay	MATERIAL PROPERTIES CAM_CLAY MODIFIED (Elastic: Isotropic, Plastic)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	MATERIAL PROPERTIES NONLINEAR 81 (Volumetric Crushing or Crushable Foam)
	Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)

Creep	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic		VISCO ELASTIC PROPERTIES
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable	
Generic Polymer	Isotropic	MATERIAL PROPERTIES NONLINEAR 89 (Generic Polymer Model)
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V: at nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py: force per unit radian at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not available.
	FLD	Face loads . Px, Py: local face pressures at nodes (force per unit area).
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Ω_x , Ω_y (angular velocity must be applied about axis of symmetry), 0, 0.
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, ϕ_4 , Xcbf, Ycbf
Velocities	VELO	Velocities. Vx, Vy: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. σ_x , σ_y , σ_{xy} , σ_z : global stresses. ϵ_x , ϵ_y , γ_{xy} , ϵ_z : global strains.
	SSIG	Initial stresses/strains at Gauss points. σ_x , σ_y , σ_{xy} , σ_z : global stresses. ϵ_x , ϵ_y , γ_{xy} , ϵ_z : global strains.

Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σ_x , σ_y , σ_{xy} , σ_z : global stresses.
	SSRG	Residual stresses at Gauss points. σ_x , σ_y , σ_{xy} , σ_z : global stresses.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Stress (default): σ_x , σ_y , σ_{xy} , σ_z , $\sigma_{\max}$, $\sigma_{\min}$, β , σ_s , σ_e (see description of principal stresses)	
	Strain: ϵ_x , ϵ_y , γ_{xy} , ϵ_z , $\epsilon_{\max}$, $\epsilon_{\min}$, β , ϵ_s , ϵ_e	
Modeller	See Results Tables (Appendix K) .	

Local Axes

Not applicable (global axes are the reference).

Sign Convention

- ☐ [Standard 2D continuum element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	For large displacements and large rotations.
Updated Lagrangian	For large displacements and large rotations.
Eulerian	For large displacements, large rotations and moderately large strains.
Co-rotational	Not applicable.

Integration Schemes

Stiffness	Default.	1-point (TAX3), 3-point (TAX6), 2x2 (QAX4, QAX8)
	Fine (see <i>Options</i>).	3x3 (QAX8), 3-point (TAX3).
Mass	Default.	1-point (TAX3), 3-point (TAX6), 2x2 (QAX4, QAX8)

Fine (see *Options*). 3x3 (QAX8), 3-point (TAX3).

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes finer integration rule.
- 47** X-axis taken as axis of symmetry
- 54** Updated Lagrangian geometric nonlinearity.
- 55** Output strains as well as stresses.
- 87** Total Lagrangian geometric nonlinearity.
- 91** Invokes fine integration rule for mass matrix.
- 105** Lumped mass matrix.
- 123** Clockwise node numbering.
- 139** Output yielded Gauss points only.
- 167** Eulerian geometric nonlinearity.

Notes on Use

1. The element formulations are based on the standard [isoparametric](#) approach. The variation of stresses within an element can be regarded as constant for the lower order (corner node only) elements, and linear for the higher order (mid-side node) elements.
2. All elements pass the [patch test](#).
3. Non-conservative loading is available with these elements when using either Updated Lagrangian or Eulerian geometric nonlinear formulations together with the FLD loading facility.
4. Option 123 will not operate on a mesh with a mixture of clockwise and anti-clockwise elements, it is only applicable if **every** element is numbered **clockwise**. Surface normals should be visualised and if necessary corrected in the pre-processing stage.
5. Using Option 123 with local loading types, such as FLD and UDL, will cause load reversal.
6. The maximum and minimum principal stress computations for axisymmetric elements do not include the σ_z term as this is implicitly a principal stress in a biaxial stress field.

Restrictions

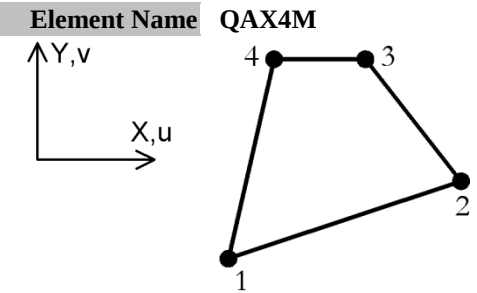
- ☐ [Ensure mid-side node centrality](#)
- ☐ [Avoid excessive element curvature](#)
- ☐ Avoid excessive aspect ratio

Recommendations on Use

- The 8-noded element with a 2*2 Gauss rule is usually the most effective element, as the under-integration of the stiffness matrix prevents locking, which may occur either when the element is subjected to [parasitic shear](#), or as the material reaches the incompressible limit (elasto-plasticity). The Gauss point stresses are also sampled at the most accurate locations for the element. However, the element does possess one spurious zero energy mode. This mode is very rarely activated in linear analysis, but it may occur in both materially and geometrically nonlinear analyses. Therefore, a careful examination of the solution should be performed, to check for spurious stress oscillations and peculiarities in the deformed configuration.
- The 8-noded element with a 3*3 Gauss rule may be used if a spurious mechanism is excited with the 2*2 Gauss rule.
- The 4-noded element should not be used for analyses where in-plane bending effects are significant as the element tends to lock in [parasitic shear](#).

2D Axisymmetric Solid Continuum Element with Enhanced Strains

General



Element Group	2D Continuum
Element Subgroup	Axisymmetric Solid
Element Description	A 2D isoparametric element with an assumed strain field. This mixed assumed strain element demonstrates a superior performance to QAX4 (see Notes). The formulations apply over a unit radian segment of the structure, and the loading and boundary conditions are axisymmetric. By default, the Y-axis is taken as the axis of symmetry. The element is numerically integrated.
Number Of Nodes	4, numbered anticlockwise.
Freedom	U, V: at each node.
Node Coordinates	X, Y: at each node.

Geometric Properties

Not applicable (a unit radian segment is assumed).

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC AXISYMMETRIC (Elastic: Orthotropic Axisymmetric)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC 4 (Not supported in LUSAS Modeller)
	Rigidities.	Not applicable

Matrix	Not applicable	
Joint	Not applicable	
Concrete		MATERIAL PROPERTIES NONLINEAR 94 (Elastic: Isotropic, Plastic: Multi-Crack Concrete) MATERIAL PROPERTIES NONLINEAR 102 (Elastic: Isotropic, Plastic: Smoothed Multi-Crack Concrete)
Elasto-Plastic	Stress resultant:	Not applicable.
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Modified Mohr-Coulomb:	MATERIAL PROPERTIES MODIFIED MOHR_COULOMB (Elastic: Isotropic, Plastic: Mohr-Coulomb/Tresca, non-associative Hardening with tension/compression cut-off)
	Modified Cam-clay	MATERIAL PROPERTIES CAM_CLAY MODIFIED (Elastic: Isotropic, Plastic)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	MATERIAL PROPERTIES NONLINEAR 81 (Volumetric Crushing or Crushable Foam)
	Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
Creep		CREEP PROPERTIES (Creep)
	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB=FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)

	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic		VISCO ELASTIC PROPERTIES
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable	
Generic	Isotropic	MATERIAL PROPERTIES NONLINEAR 89 (Generic Polymer Model)
Polymer		
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V: at nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py: force per unit radian at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not available.
	FLD	Face loads. Px, Py: local face pressures at nodes (force per unit area).
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Ω_x , Ω_y (angular velocity must be applied about axis of symmetry), 0,0.
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, ϕ_4 , Xcbf, Ycbf
Velocities	VELO	Velocities. Vx, Vy: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. σ_x , σ_y , σ_{xy} , σ_z : global stresses. ϵ_x , ϵ_y , γ_{xy} , ϵ_z : global strains.
	SSIG	Initial stresses/strains at Gauss points. σ_x , σ_y , σ_{xy} , σ_z : global stresses. ϵ_x , ϵ_y , γ_{xy} , ϵ_z : global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σ_x , σ_y , σ_{xy} , σ_z : global stresses.
	SSRG	Residual stresses at Gauss points. σ_x , σ_y , σ_{xy} , σ_z : global stresses.

Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Stress (default): σ_x , σ_y , σ_{xy} , σ_z , σ_{max} , σ_{min} , β , σ_s , σ_e (see description of principal stresses)
	Strain: ϵ_x , ϵ_y , γ_{xy} , ϵ_z , ϵ_{max} , ϵ_{min} , β , ϵ_s , ϵ_e
Modeller	See Results Tables (Appendix K) .

Local Axes

Not applicable (global axes are the reference).

Sign Convention

- ☐ [Standard 2D continuum element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	For large displacements and large rotations.
Updated Lagrangian	For large displacements and large rotations.
Eulerian	For large displacements, large rotations and moderately large strains.
Co-rotational	Not applicable.

Integration Schemes

Stiffness	Default.	2x2
	Fine.	As default.
Mass	Default.	2x2
	Fine.	As default.

Mass Modelling

- ☐ Consistent mass (default).

- ☐ Lumped mass.

Options

- 47 X-axis taken as axis of symmetry
- 54 Updated Lagrangian geometric nonlinearity
- 55 Output strains as well as stresses.
- 87 Total Lagrangian geometric nonlinearity.
- 91 Invokes fine integration rule for mass matrix.
- 105 Lumped mass matrix.
- 123 Clockwise node numbering.
- 139 Output yielded Gauss points only.
- 167 Eulerian geometric nonlinearity.

Notes on Use

1. The element formulations are based on the standard [isoparametric](#) approach. The variation of stresses within an element can be regarded as linear.
2. All elements pass the [patch test](#).
3. The strain field for this element consists of two parts: the compatible strains derived from an assumed displacement field and the assumed enhanced strains; see *LUSAS Theory Manual*. The assumed enhanced strain field is defined using 5 parameters for both linear and nonlinear applications.
4. Non-conservative loading is available with these elements when using either Updated Lagrangian or Eulerian geometric nonlinear formulations together with the FLD loading facility.
5. Option 123 will not operate on a mesh with a mixture of clockwise and anti-clockwise elements, it will only work if **every** element is numbered **clockwise**. The best way to avoid a mixture is to check and appropriately reverse the surface definitions in the pre-processing stage of modelling.
6. Using Option 123 with local loading types, such as FLD and UDL, will cause load reversal.
7. The maximum and minimum principal stress computations for axisymmetric elements do not include the σ_z term as this is implicitly a principal stress in a biaxial stress field.
8. Convergence difficulties can sometimes arise when using enhanced strain elements with nonlinear materials, particularly if the material is elastic-perfectly plastic or if a very shallow hardening curve is defined. In such cases it is recommended that the standard element formulation is used.
9. This element exhibits an improved performance when compared with its parent element QAX4. The integration rules are the same as the parent element. The

elements do not suffer from locking due to [parasitic shear](#) or when the material approaches the incompressible limit. The elements are also free of any [zero energy modes](#).

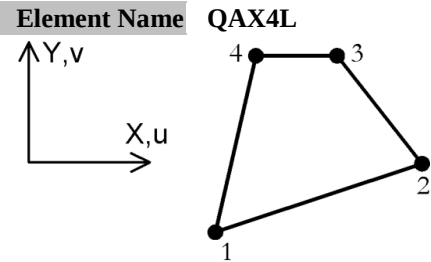
10. In analyses where significant in-plane bending is thermally induced it is recommended that a nonlinear solution is used. If a linear solution is required, then quadratic plane strain elements QPN8 are recommended.

Restrictions

- ❑ [Avoid excessive aspect ratio](#)

2D Axisymmetric Solid Continuum Element for Large Strains

General



Element Group	2D Continuum
Element Subgroup	Axisymmetric Solid
Element Description	A 2D isoparametric element incorporating an internal pressure variable. This element should be used for analyses involving large strains. The formulations apply over a unit radian segment of the structure and the loading and boundary conditions are axisymmetric. By default, the Y-axis is taken as the axis of symmetry. The element is numerically integrated.
Number Of Nodes	4, numbered anticlockwise.
Freedom	U, V: at each node.
Node Coordinates	X, Y: at each node.

Geometric Properties

Not applicable (a unit radian segment is assumed).

Material Properties

Linear	Not applicable	
Matrix	Not applicable	
Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic	Implicit	MATERIAL PROPERTIES NONLINEAR 75
	Optimised Von Mises	(Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic)
	Stress Potential	STRESS POTENTIAL VON_MISES (Isotropic:

		von Mises)
Creep	Not applicable	
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage	Not applicable	
Rubber	Ogden	MATERIAL PROPERTIES RUBBER OGDEN (Rubber: Ogden)
	Mooney-Rivlin	MATERIAL PROPERTIES RUBBER MOONEY_RIVLIN (Rubber: Mooney-Rivlin)
	Neo-Hookean	MATERIAL PROPERTIES RUBBER NEO_HOOKEAN (Rubber: Neo-Hookean)
	Hencky	MATERIAL PROPERTIES RUBBER HENCKY (Rubber: Hencky)
Generic Polymer	Not applicable	
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V: at nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py: force per unit radian at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not available.
	FLD	Face loads . Px, Py: local face pressures at nodes (force per unit area).
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Ω_x , Ω_y , (angular velocity must be applied about axis of symmetry), 0,0.
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, ϕ^4 , Xcbf, Ycbf
Velocities	VELO	Velocities. Vx, Vy: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. σ_x , σ_y , σ_{xy} , σ_z : global stresses. ϵ_x , ϵ_y , γ_{xy} , ϵ_z : global strains.
	SSIG	Initial stresses/strains at Gauss points. σ_x , σ_y , σ_{xy} , σ_z : global stresses. ϵ_x , ϵ_y , γ_{xy} , ϵ_z : global strains.

Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σ_x , σ_y , σ_{xy} , σ_z : global stresses.
	SSRG	Residual stresses at Gauss points. σ_x , σ_y , σ_{xy} , σ_z : global stresses.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver Stress (default): σ_x , σ_y , σ_{xy} , σ_z , $\sigma_{\max}$, $\sigma_{\min}$, β , σ_s , σ_e (see [description of principal stresses](#))

Principal stretches, λ_1 , λ_2 , λ_{31} , $\theta\lambda$, $\det F$. Where λ_i are the principal stretches, $\theta\lambda$ the angle between the maximum principal stretch and the global X axis, and $\det F$ the determinant of the deformation gradient or volume ratio.

Modeller See [Results Tables \(Appendix K\)](#).

Local Axes

Not applicable (global axes are the reference).

Sign Convention

☐ [Standard 2D continuum element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	Not applicable.
Updated Lagrangian	Not applicable.
Eulerian	For large displacements and large strains.
Co-rotational	Not applicable.

Integration Schemes

Stiffness	Default.	2x2
	Fine.	As default.

Mass Default. 2x2
Fine. As default.

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 47 X-axis taken as axis of symmetry.
- 55 Output stretches as well as stresses.
- 91 Invokes fine integration rule for mass matrix.
- 105 Lumped mass matrix
- 123 Clockwise node numbering.

Notes on Use

1. The element formulations are based on the standard [isoparametric](#) approach. The variation of stresses within an element can be regarded as linear.
2. The element passes the large strain patch test for rubber.
3. Non-conservative loading is available with this element when using FLD loading.
4. The stresses output are [Kirchhoff](#) stresses (see *LUSAS Theory Manual*).
5. Stretch output consists of the principal stretches and the angle defining the principal directions. The value of $\det F = \lambda_1 \lambda_2$ is also output. (Refer to the *LUSAS Theory Manual* for more details.)
6. Option 123 will not operate on a mesh with a mixture of clockwise and anti-clockwise elements, it will only work if **every** element is numbered **clockwise**. The best way to avoid a mixture is to check and appropriately reverse the surface definitions in the pre-processing stage of modelling.
7. Using Option 123 with local loading types, such as FLD and UDL, will cause load reversal.
8. The maximum and minimum principal stress computations for axisymmetric elements do not include the σ_z term as this is implicitly a principal stress in a biaxial stress field.

Restrictions

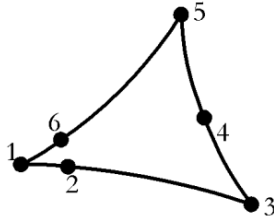
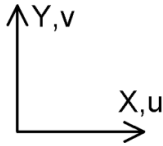
- ☐ Avoid excessive aspect ratio

- ❑ Avoid non-uniform initial and thermal strains with coarse meshes

2D Axisymmetric Solid Continuum Crack Tip Elements

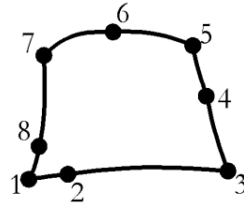
General

Element Name TXK6



Crack specified at Node 1

Element Name QXK8



Crack specified at Node 1

Element Group 2D Continuum

Element Subgroup Axisymmetric Solid

Element Description

A family of 2D [isoparametric](#) crack tip elements where the crack tip can be located at any node. The mid-side nodes are moved to the quarter points to produce a singularity at the crack tip. The strains vary as the square root of $1/R$, where R is the distance from the crack tip. These elements are used at the crack tip only and should be mixed with the higher order axisymmetric solid continuum elements. The formulations apply over a unit radian segment of the structure, and the loading and boundary conditions are axisymmetric. By default, the Y-axis is taken as the axis of symmetry. The elements are numerically integrated.

Number Of Nodes

6 or 8 numbered anticlockwise.

Freedom

U, V: at each node.

Node

X, Y: at each node.

Coordinates

Geometric Properties

Not applicable (a unit radian segment is assumed).

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC AXISYMMETRIC (Elastic: Orthotropic Axisymmetric)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC 4 (Not

		supported in LUSAS Modeller)
	Rigidities.	Not applicable.
Matrix	Not applicable	
Joint	Not applicable	
Concrete		MATERIAL PROPERTIES NONLINEAR 94 (Elastic: Isotropic, Plastic: Multi-Crack Concrete)
		MATERIAL PROPERTIES NONLINEAR 102 (Elastic: Isotropic, Plastic: Smoothed Multi-Crack Concrete)
Elasto-Plastic	Stress resultant:	Not applicable.
	Interface:	MATERIAL PROPERTIES NONLINEAR 27
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Modified Mohr-Coulomb:	MATERIAL PROPERTIES MODIFIED MOHR_COULOMB (Elastic: Isotropic, Plastic: Mohr-Coulomb/Tresca, non-associative Hardening with tension/compression cut-off)
	Modified Cam-clay	MATERIAL PROPERTIES CAM_CLAY MODIFIED (Elastic: Isotropic, Plastic)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	MATERIAL PROPERTIES NONLINEAR 81 (Volumetric Crushing or Crushable Foam)
	Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
Creep		CREEP PROPERTIES (Creep)
	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	CHINESE	MATERIAL PROPERTIES NONLINEAR 86 CHINESE

		(Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE
		(Concrete creep model to EUROCODE_2)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic		VISCO ELASTIC PROPERTIES
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable	
Generic	Isotropic	MATERIAL PROPERTIES NONLINEAR 89
Polymer		(Generic Polymer Model)
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V: at nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py: at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	Face loads . Px, Py: local face axis pressures at nodes.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Ω_x , Ω_y (angular velocity must be applied about axis of symmetry), 0, 0.
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, ϕ_4 , Xcbf, Ycbf
Velocities	VELO	Velocities. Vx, Vy: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. σ_x , σ_y , σ_{xy} , σ_z : global stresses. ϵ_x , ϵ_y , γ_{xy} , ϵ_z : global strains.
	SSIG	Initial stresses/strains at Gauss points. σ_x , σ_y , σ_{xy} , σ_z : global stresses. ϵ_x , ϵ_y , γ_{xy} , ϵ_z : global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σ_x , σ_y , σ_{xy} , σ_z : global stresses.
	SSRG	Residual stresses at Gauss points. σ_x , σ_y , σ_{xy} , σ_z : global stresses.

Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Stress (default): σ_x , σ_y , σ_{xy} , σ_z , σ_{max} , σ_{min} , β , σ_s , σ_e (see description of principal stresses)	
	Strain: ϵ_x , ϵ_y , γ_{xy} , ϵ_z , ϵ_{max} , ϵ_{min} , β , ϵ_s , ϵ_e	
Modeller	See Results Tables (Appendix K) .	

Local Axes

Not applicable (global axes are the reference).

Sign Convention

- ☐ [Standard 2D continuum element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	For large displacements and large rotations.	
Updated Lagrangian	For large displacements and large rotations.	
Eulerian	For large displacements, large rotations and moderately large strains.	
Co-rotational	Not applicable.	

Integration Schemes

Stiffness	Default.	6-point (TXK6), 3x3 (QXK8)
	Fine (see <i>Options</i>).	12-point (TXK6).
Mass	Default.	6-point (TXK6), 3x3 (QXK8)
	Fine (see <i>Options</i>).	12-point (TXK6).

Mass Modelling

- ☐ Consistent mass (default).

- ☐ Lumped mass.

Options

- 18** Invokes finer integration rule.
- 47** X-axis taken as axis of symmetry.
- 54** Updated Lagrangian geometric nonlinearity.
- 55** Output strains as well as stresses.
- 87** Total Lagrangian geometric nonlinearity.
- 91** Invokes fine integration rule for mass matrix.
- 105** Lumped mass matrix
- 123** Clockwise node numbering.
- 139** Output yielded Gauss points only.
- 167** Eulerian geometric nonlinearity.

Notes on Use

- The element formulations are based on the standard [isoparametric](#) approach. Moving the mid-side nodes to the quarter points creates a singularity with theoretically infinite stress at the corner node.
- Non-conservative loading is available with these elements when using either Updated Lagrangian or Eulerian geometric nonlinear formulations together with the FLD loading facility.
- Option 123 will not operate on a mesh with a mixture of clockwise and anti-clockwise elements, it will only work if **every** element is numbered **clockwise**. The best way to avoid a mixture is to check and appropriately reverse the surface definitions in the pre-processing stage of modelling.
- Using Option 123 with local loading types, such as FLD and UDL, will cause load reversal.
- The maximum and minimum principal stress computations for axisymmetric elements do not include the σ_z term as this is implicitly a principal stress in a biaxial stress field.

Restrictions

- ☐ [Avoid excessive element curvature](#)
- ☐ Avoid excessive aspect ratio

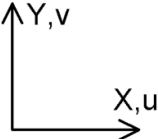
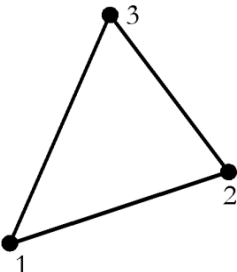
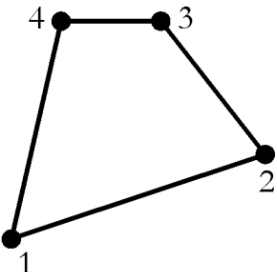
Recommendations on Use

The QXK8 and TXK6 elements are specifically designed for application to fracture mechanics problems and may be used to model the singularities that occur at the crack tip. The mid-side nodes near the crack tip are shifted to the quarter point. This ensures a singularity is present at the crack tip and that the strains vary as $1/\sqrt{r}$.

where r is the distance from the crack tip. The triangular TPK6 element is more effective than the quadrilateral element.

2D Axisymmetric Solid Explicit Dynamics Elements

General

Element Name	TAX3E	QAX4E
		
Element Group	2D Continuum	
Element Subgroup	Axisymmetric Solid Continuum	
Element Description	A family of 2D isoparametric elements for explicit dynamic analyses. The formulations apply over a unit radian segment of structure and loading boundary conditions are axisymmetric. By default, the Y-axis is taken as the axis of symmetry. The elements are numerically integrated.	
Number Of Nodes	3 or 4 numbered anticlockwise.	
Freedom Node Coordinates	U, V: at each node. X, Y: at each node.	

Geometric Properties

Not applicable (a unit radian segment is assumed).

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC AXISYMMETRIC (Elastic: Orthotropic Axisymmetric)
	Anisotropic:	Not applicable
	Rigidities.	Not applicable
Matrix	Not applicable	
Joint	Not applicable	
Concrete	Not applicable	

Elasto-Plastic	Stress resultant:	Not applicable
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Modified Mohr-Coulomb:	MATERIAL PROPERTIES MODIFIED MOHR_COULOMB (Elastic: Isotropic, Plastic: Mohr-Coulomb/Tresca, non-associative Hardening with tension/compression cut-off)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	MATERIAL PROPERTIES NONLINEAR 81 (Volumetric Crushing or Crushable Foam)
	Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
Creep		CREEP PROPERTIES (Creep) (See <i>Notes</i>)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic		VISCO ELASTIC PROPERTIES
Shrinkage	Not applicable	
Rubber	Not applicable	
Generic Polymer	Not applicable	
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V: at each node.
Concentrated Loads	CL	Concentrated loads. Px, Py: at each node.
Element Loads	Not applicable.	
Distributed	UDL	Not applicable.

	Loads	FLD	Face loads . Px, Py: local face axis pressures at nodes.
	Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Ω_x , Ω_y (angular velocity must be applied about axis of symmetry), 0,0.
		BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, ϕ_4 , Xcbf, Ycbf
	Velocities	VELO	Velocities. Vx, Vy at nodes.
	Accelerations	ACCE	Acceleration. Ax, Ay at nodes.
Initial Stress/Strains	Initial	SSI, SSIE	Initial stresses/strains at nodes/for element.
			σ_x , σ_y , σ_{xy} , σ_z : global stresses. ϵ_x , ϵ_y , γ_{xy} , ϵ_z : global strains.
		SSIG	Initial stress/strains at Gauss points. σ_x , σ_y , σ_{xy} , σ_z : global stress. ϵ_x , ϵ_y , γ_{xy} , ϵ_z : global strains.
Residual Stresses		SSR, SSRE	Residual stresses at nodes/for element σ_x , σ_y , σ_{xy} , σ_z : global stresses.
		SSRG	Residual stresses at Gauss points. σ_x , σ_y , σ_{xy} , σ_z : global stresses.
Temperatures		TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads		Not applicable.	
Temp Dependent Loads		Not applicable.	

LUSAS Output

Solver	Stress (default): σ_x , σ_y , σ_{xy} , σ_z , $\sigma_{\max}$, $\sigma_{\min}$, β , σ_s , σ_e (see description of principal stresses)
	Strain: ϵ_x , ϵ_y , γ_{xy} , ϵ_z , $\epsilon_{\max}$, $\epsilon_{\min}$, β , ϵ_s , ϵ_e
Modeller	See Results Tables (Appendix K)

Local Axes

Not applicable.

Sign Convention

- ☐ Standard 2D continuum element

Formulation

Geometric Nonlinearity

Total Lagrangian	Not applicable.
Updated Lagrangian	Not applicable.
Eulerian	For large displacements, large rotations and moderately large strains.
Co-rotational	Not applicable.

Integration Schemes

Stiffness	Default.	1-point (see <i>Notes</i>)
	Fine.	As default.
Mass	Default.	1-point (see <i>Notes</i>)
	Fine.	As default.

Mass Modelling

- ☐ Lumped mass (see *Notes*).

Options

- 47** X-axis taken as axis of symmetry
- 55** Output strains as well as stresses.
- 105** Lumped mass matrix (see *Notes*).
- 139** Output yielded Gauss points only.

Notes on Use

- The element formulations are based on the standard
- The system parameter HGVISC is used to restrict element mechanisms due to under-integration. The default value is usually sufficient.
- The bulk viscosity coefficients are used to restrict numerical oscillations due to the traversal of stress waves. The default bulk viscosity coefficients (BULKLF and BULKQF) may be altered as a SYSTEM parameter.
- These elements must be used with a dynamic central difference scheme and a lumped mass matrix.
- These elements are not applicable to static or eigenvalue analyses.

14. Automatic time step calculations are implemented.
15. As the element geometry is always updated in an explicit dynamic analysis, a nonlinear solution is obtained. When using explicit dynamics elements Nonlinear Control must be specified.
16. If CREEP PROPERTIES are defined explicit time integration must be specified in VISCOUS CONTROL.
17. Non-conservative loading is invoked when the face loading (FLD) is applied.
18. Rayleigh damping coefficients are not supported by these elements.
19. Constraint equations are not available for use with these elements.
20. Nodes **must** be specified in an anticlockwise order. Option 123 is **not** applicable for this element. When using Modeller ensure surface normal is in the +ve z direction.
21. The maximum and minimum principal stress computations for axisymmetric elements do not include the σ_z term as this is implicitly a principal stress in a biaxial stress field.

Restrictions

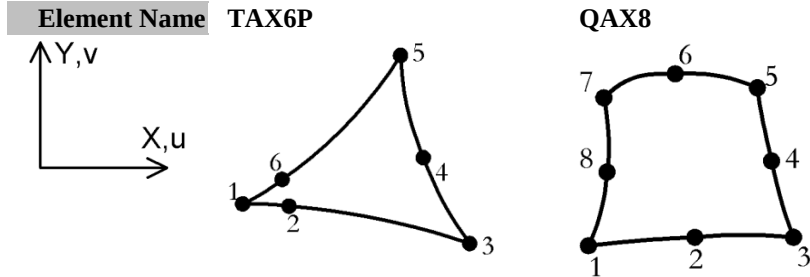
- ❑ [Avoid excessive aspect ratio](#)

Recommendations on Use

Explicit dynamics elements may be used to define surface boundaries which will be active in a slideline analysis.

2D Axisymmetric Solid Two Phase Continuum Elements

General



Element Group	2D Continuum
Element Subgroup	Axisymmetric Solid
Element Description	A family of 2D isoparametric elements with higher order models capable of modelling curved boundaries. The formulations apply over a unit radian segment of the structure and the loading and boundary conditions are axisymmetric. By default, the Y-axis is taken as the axis of symmetry. The elements are numerically integrated.
Number Of Nodes	6 or 8 numbered anticlockwise.
Freedom Node Coordinates	U, V, P: at corner nodes. U, V: at midside nodes. X, Y: at each node.

Geometric Properties

Not applicable (a unit radian segment is assumed).

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC AXISYMMETRIC (Elastic: orthotropic, Axisymmetric)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC 4 (Not supported in LUSAS Modeller)
	Rigidities.	Not applicable.
Matrix	Not applicable	

Joint	Not applicable	
Concrete		MATERIAL PROPERTIES NONLINEAR 94 (Elastic: Isotropic, Plastic: Multi-Crack Concrete) MATERIAL PROPERTIES NONLINEAR 102 (Elastic: Isotropic, Plastic: Smoothed Multi-Crack Concrete)
Elasto-Plastic	Stress resultant:	Not applicable.
	Interface:	MATERIAL PROPERTIES NONLINEAR 27.
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Modified Mohr-Coulomb:	MATERIAL PROPERTIES MODIFIED MOHR_COULOMB (Elastic: Isotropic, Plastic: Mohr-Coulomb/Tresca, non-associative Hardening with tension/compression cut-off)
	Modified Cam-clay	MATERIAL PROPERTIES CAM_CLAY MODIFIED (Elastic: Isotropic, Plastic)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	MATERIAL PROPERTIES NONLINEAR 81 (Volumetric Crushing or Crushable Foam)
	Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
Creep	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE

Damage		(Concrete creep model to EUROCODE_2)
Viscoelastic		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Shrinkage		VISCO ELASTIC PROPERTIES SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable	
Generic Polymer	Isotropic	MATERIAL PROPERTIES NONLINEAR 89 (Generic Polymer Model)
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, P: at corner nodes. U, V: at midside nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py, Q: force/flux per unit radian at corner nodes. Px, Py: force per unit radian at midside nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not available.
	FLD	Face loads . Px, Py, Q: local face pressures/flux at corner nodes (force/flux per unit area). Px, Py: local face pressures at midside nodes.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Ω_x , Ω_y (angular velocity must be applied about axis of symmetry), 0, 0, gx, gy. (See Notes on Use)
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, ϕ_4 , Xcbf, Ycbf, gx, gy. (See Notes on Use)
Velocities	VELO	Velocities. Vx, Vy: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. σ_x , σ_y , σ_{xy} , σ_z , σ_p : global stresses. ϵ_x , ϵ_y , γ_{xy} , ϵ_z : global strains.
	SSIG	Initial stresses/strains at Gauss points. σ_x , σ_y , σ_{xy} , σ_z , σ_p : global stresses. ϵ_x , ϵ_y , γ_{xy} , ϵ_z : global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σ_x , σ_y , σ_{xy} , σ_z , σ_p : global stresses.
	SSRG	Residual stresses at Gauss points. σ_x , σ_y , σ_{xy} , σ_z , σ_p : global stresses.

Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Stress (default): σ_x , σ_y , σ_{xy} , σ_z , σ_p , $\sigma_{\max}$, $\sigma_{\min}$, β , σ_s , σ_e (see description of principal stresses) Strain: ϵ_x , ϵ_y , γ_{xy} , ϵ_z , $\epsilon_{\max}$, $\epsilon_{\min}$, β , ϵ_s , ϵ_e
Modeller	See Results Tables (Appendix K) .

Local Axes

Not applicable (global axes are the reference).

Sign Convention

- ☐ [Standard 2D continuum element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	For large displacements and large rotations.
Updated Lagrangian	For large displacements and large rotations.
Eulerian	For large displacements, large rotations and moderately large strains.
Co-rotational	Not applicable.

Integration Schemes

Stiffness	Default.	3-point (TAX6P), 2x2 (QAX8P)
	Fine (see <i>Options</i>).	3x3 (QAX8P)
Mass	Default.	3-point (TAX6P), 2x2 (QAX8P)
	Fine (see <i>Options</i>).	3x3 (QAX8P)

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes finer integration rule.
- 47** X-axis taken as axis of symmetry
- 54** Updated Lagrangian geometric nonlinearity.
- 55** Output strains as well as stresses.
- 87** Total Lagrangian geometric nonlinearity.
- 91** Invokes fine integration rule for mass matrix.
- 105** Lumped mass matrix.
- 123** Clockwise node numbering.
- 139** Output yielded Gauss points only.
- 167** Eulerian geometric nonlinearity.

Notes on Use

1. Two phase material parameters must be used with these elements for undrained and consolidation analysis.
2. The element formulations are based on the standard [isoparametric](#) approach. The variation of isoparametric stresses and pore pressures within an element can be regarded as linear.
3. All elements pass the [patch test](#).
4. Non-conservative loading is available with these elements when using either Updated Lagrangian or Eulerian geometric nonlinear formulations together with the FLD loading facility.
5. Option 123 will not operate on a mesh with a mixture of clockwise and anti-clockwise elements, it is only applicable if **every** element is numbered **clockwise**. Surface normals should be visualised and if necessary corrected in the pre-processing stage.
6. Using Option 123 with local loading types, such as FLD and UDL, will cause load reversal.
7. The global components of gravity acting on the fluid phase are defined by g_x and g_y under CBF and BFP loading.
8. The maximum and minimum principal stress computations for axisymmetric elements do not include the σ_z term as this is implicitly a principal stress in a biaxial stress field.

Restrictions

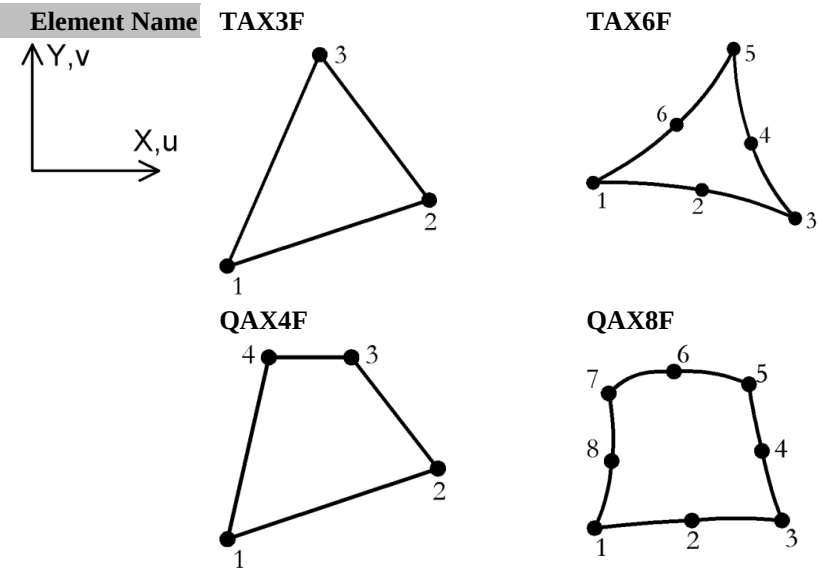
- ☐ [Ensure mid-side node centrality](#)
- ☐ [Avoid excessive element curvature](#)
- ☐ Avoid excessive aspect ratio

Recommendations on Use

- The 8-noded element with a 2*2 Gauss rule is usually the most effective element, as the under-integration of the stiffness matrix prevents locking, which may occur either when the element is subjected to [parasitic shear](#), or as the material reaches the incompressible limit (elasto-plasticity). The Gauss point stresses are also sampled at the most accurate locations for the element. However, the element does possess one spurious zero energy mode. This mode is very rarely activated in linear analysis, but it may occur in both materially and geometrically nonlinear analyses. Therefore, a careful examination of the solution should be performed, to check for spurious stress oscillations and peculiarities in the deformed configuration.
- The 8-noded element with a 3*3 Gauss rule may be used if a spurious mechanism is excited with the 2*2 Gauss rule.

2D Axisymmetric Fourier Ring Elements

General



Element Group	2D Continuum
Element Subgroup	Fourier Ring
Element Description	A family of 2D isoparametric elements with higher order models capable of modelling curved boundaries. The structure must be axisymmetric but the loading need not be. By default the Y-axis is taken to be the axis of symmetry. The elements are numerically integrated.
Number Of Nodes	3, 4, 6 or 8 numbered anticlockwise.
Freedom	U, V, W: at each node (in cylindrical coordinates, see local coordinates).
Node Coordinates	X, Y: at each node.

Geometric Properties

Not applicable.

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC (Elastic: Orthotropic Plane Stress)
		MATERIAL PROPERTIES ORTHOTROPIC SOLID (Elastic: Orthotropic Solid)
	Anisotropic:	Not applicable
	Rigidities.	Not applicable
Matrix	Not applicable	
Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic	Not applicable	
Creep	Not applicable	
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage	Not applicable	
Rubber	Not applicable	
Generic	Not applicable	
Polymer		
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W: at each node.
Concentrated Loads	CL	Concentrated loads. Px, Py, Pz: at each node (global, may also be applied locally, see options).
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	Face loads . Px, Py, Pz: local face axis pressures at nodes Pz in the direction of increasing θ .
Body Forces	CBF	Constant body forces for element (see <i>Notes</i>). Xcbf, Ycbf, Zcbf, Ω_x , Ω_y , Ω_z , α_x , α_y , α_z , Xo, Yo, Zo, $d\theta/dt$
	BFP, BFPE	Body force potentials at nodes/for element. Xcbf, Ycbf, Zcbf
Velocities	VELO	Velocities. Vx, Vy, Vz at nodes.
Accelerations	ACCE	Acceleration. Ax, Ay, Az at nodes.
Initial	SSI, SSIE	Initial stresses/strains at nodes/for element. σ_x ,

Stress/Strains		$\sigma_y, \sigma_z, \sigma_{xy}, \sigma_{yz}, \sigma_{xz}$: local stresses. $\epsilon_x, \epsilon_y, \epsilon_z, \gamma_{xy}, \gamma_{yz}, \gamma_{xz}$: local strains.
	SSIG	Initial stresses/strains at Gauss points. $\sigma_x, \sigma_y, \sigma_z, \sigma_{xy}, \sigma_{yz}, \sigma_{xz}$: local stresses. $\epsilon_x, \epsilon_y, \epsilon_z, \gamma_{xy}, \gamma_{yz}, \gamma_{xz}$: local strains.
Residual Stresses	Not applicable.	
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp	Not applicable.	
Dependent Loads		

LUSAS Output

Solver	Stress (default): $\sigma_x, \sigma_y, \sigma_z, \sigma_{xy}, \sigma_{yz}, \sigma_{xz}, \sigma_{max}, \sigma_{min}, \beta, \sigma_s, \sigma_e$ (see description of principal stresses) Strain: $\epsilon_x, \epsilon_y, \epsilon_z, \gamma_{xy}, \gamma_{yz}, \gamma_{xz}, \epsilon_{max}, \epsilon_{min}, \beta, \epsilon_s, \epsilon_e$ Use LUSAS Modeller to access results at various angles around the structure. See Local and Global Results in the <i>Modeller User Manual</i>
Modeller	See Results Tables (Appendix K) .

Local Axes

- ☐ Cylindrical coordinates (see *Appendix F*).
- ☐ The element axes are defined in the cylindrical coordinate system x,y,z, with associated displacements u,v,w. The tangential displacement w is positive in the direction of increasing θ , where θ is the positive rotation defined by the right-hand coordinate system about the axis of symmetry. u and v are positive in the direction of increasing x and y respectively and may be either axial or radial displacements depending on the definition of the axis of symmetry.

Sign Convention

- ☐ Standard 3D continuum element

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Stiffness	Default.	1-point (TAX3F), 3-point (TAX6F), 2x2 (QAX4F, QAX8F)
	Fine (see <i>Options</i>).	3x3 (QAX8F), 3-point (TAX3F)
Mass	Default.	1-point (TAX3F), 3-point (TAX6F), 2x2 (QAX4F, QAX8F)
	Fine (see <i>Options</i>).	3x3 (QAX8F), 3-point (TAX3F)

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes fine integration rule.
- 47** X-axis taken as axis of symmetry.
- 55** Output strains as well as stresses.
- 102** Switch off load correction stiffness matrix due to centripetal acceleration.
- 105** Lumped mass matrix.
- 202** Apply concentrated loads in cylindrical coordinates.

Notes on Use

1. CBF loads are always applied as acceleration loading. X_o , Y_o , Z_o , permit a shift in the original point of the global coordinate system (about which the rotations are applied). $d\theta/dt$ is the local angular velocity about the finite element coordinate system.
2. The application of the CBF loading depends on the particular element material model selected. See the description of Fourier analysis in *Chapter 2* of the *LUSAS User Guide*.
3. If CBF loads are used the structure must be axisymmetric about the X-axis (option 47).

4. Fourier elements cannot be mixed with other element types.
5. Temperature fields cannot be used in dynamic or harmonic response analyses.
6. Centripetal load stiffening has been applied to the $n=0$ term, but there is no nonlinear stress stiffening contribution. The centripetal load stiffening matrix, contrary, to its name, actually decreases the stiffness of the structure. Centripetal forces are proportional to the angular rotation squared and the lever arm of the mass from the centre of rotation. As the body spins, the lever arm is lengthened by positive displacements, which increases the applied load. This may, conversely, be thought of as reducing the stiffness. The centripetal load stiffness is applied by default, but it may be omitted by setting option 102.
7. The maximum and minimum principal stress computations for axisymmetric elements do not include the σ_z term as this is implicitly a principal stress in a biaxial stress field.

Restrictions

- ☐ [Ensure mid-side node centrality](#)
- ☐ [Avoid excessive element curvature](#)
- ☐ Avoid excessive aspect ratio

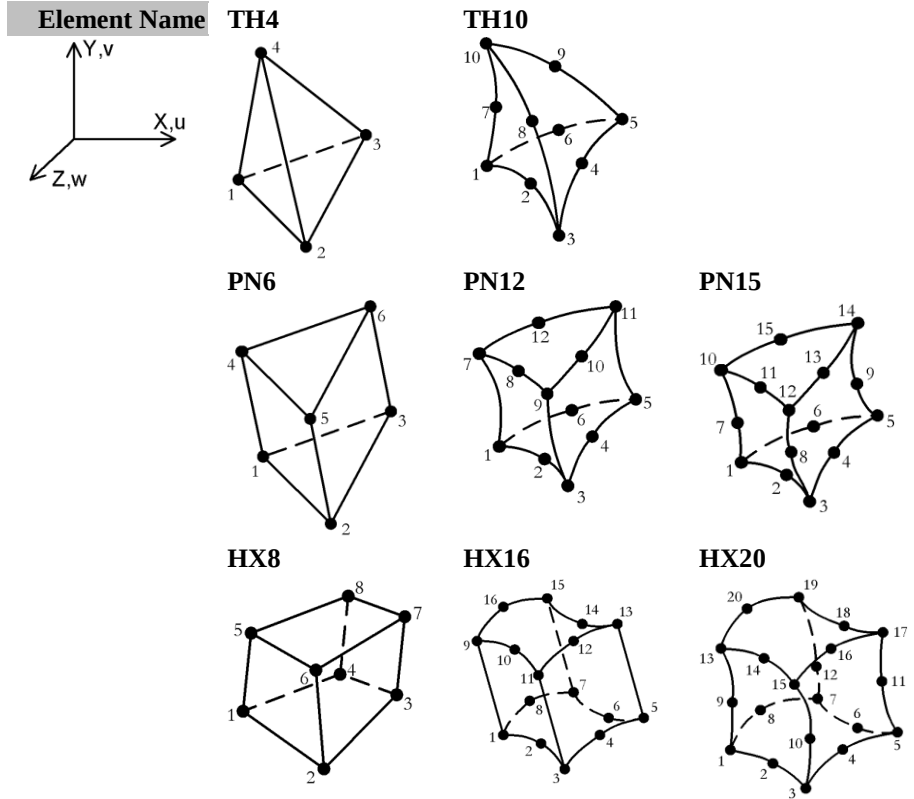
Recommendations on Use

- The element is designed to model fairly solid structures, but it also performs well in comparison to standard shell analyses and may be an effective alternative for axisymmetric problems. The QAX8F is the most effective element of the family.
- If eigenvalues are required from a thin shelled structure such as a cylinder, the Fourier elements provide an efficient means of checking a range of circumferential harmonics and will indicate the permissible coarseness of a finite element mesh which will adequately represent the 3D variation.

Chapter 4 : 3D Continuum Elements.

3D Solid Continuum Elements

General



Element Group 3D Continuum
Element Subgroup Solid Continuum

Element Description A family of 3D isoparametric solid continuum elements with higher order models capable of modelling curved boundaries. The elements are numerically integrated.

Number Of Nodes 4 or 10 (tetrahedra). 6, 12 or 15 (pentahedra). 8, 16 or 20 (hexahedra). The elements are numbered according to a right-hand screw rule in the local z-direction.

Freedom U, V, W: at each node.

Node Coordinates X, Y, Z: at each node.

Geometric Properties

Not applicable.

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC SOLID (Elastic: Orthotropic Solid)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC SOLID (Elastic: Anisotropic Solid)
	Rigidities.	Not applicable.
Matrix	Not applicable.	
Joint	Not applicable.	
Concrete		MATERIAL PROPERTIES NONLINEAR 94 (Elastic: Isotropic, Plastic: Multi-Crack Concrete)
		MATERIAL PROPERTIES NONLINEAR 102 (Elastic: Isotropic, Plastic: Smoothed Multi-Crack Concrete)
Elasto-Plastic	Stress resultant:	Not applicable.
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Modified Mohr-Coulomb:	MATERIAL PROPERTIES MODIFIED MOHR_COULOMB (Elastic: Isotropic, Plastic: Mohr-Coulomb/Tresca, non-associative Hardening with tension/compression cut-off)
	Modified Cam-clay	MATERIAL PROPERTIES CAM_CLAY MODIFIED (Elastic: Isotropic, Plastic)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	MATERIAL PROPERTIES NONLINEAR 81 (Volumetric Crushing or Crushable Foam)
	Stress Potential:	STRESS POTENTIAL VON_MISES, HILL,

		HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
Creep		CREEP PROPERTIES (Creep)
	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic		VISCO ELASTIC PROPERTIES
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Elasto-Plastic Interface		MATERIAL PROPERTIES NONLINEAR 26
Rubber	Not applicable.	
Generic Polymer	Isotropic	MATERIAL PROPERTIES NONLINEAR 89 (Generic Polymer Model)
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W: at each node.
Concentrated Loads	CL	Concentrated loads. Px, Py, Pz: at each node.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	Face Loads. Px, Py, Pz: local face pressures at nodes.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf,

		$Z_{cbf}, \Omega_x, \Omega_y, \Omega_z, \alpha_x, \alpha_y, \alpha_z$
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, ϕ_4 , X_{cbf} , Y_{cbf} , Z_{cbf}
Velocities	VELO	Velocities. V_x, V_y, V_z : at nodes.
Accelerations	ACCE	Acceleration A_x, A_y, A_z : at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. $\sigma_x, \sigma_y, \sigma_z, \sigma_{xy}, \sigma_{yz}, \sigma_{xz}$: global stresses. $\epsilon_x, \epsilon_y, \epsilon_z, \gamma_{xy}, \gamma_{yz}, \gamma_{xz}$: global strains.
	SSIG	Initial stresses/strains at Gauss points $\sigma_x, \sigma_y, \sigma_z, \sigma_{xy}, \sigma_{yz}, \sigma_{xz}$: global stresses. $\epsilon_x, \epsilon_y, \epsilon_z, \gamma_{xy}, \gamma_{yz}, \gamma_{xz}$: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. $\sigma_x, \sigma_y, \sigma_z, \sigma_{xy}, \sigma_{yz}, \sigma_{xz}$: global stresses.
	SSRG	Residual stresses at Gauss points. $\sigma_x, \sigma_y, \sigma_z, \sigma_{xy}, \sigma_{yz}, \sigma_{xz}$ global stresses.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, T_0 , 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver Stress (default): $\sigma_x, \sigma_y, \sigma_z, \sigma_{xy}, \sigma_{yz}, \sigma_{xz}, \sigma_e$: global stresses.
 Strain: $\epsilon_x, \epsilon_y, \epsilon_z, \gamma_{xy}, \gamma_{yz}, \gamma_{xz}, \epsilon_e$: global strains.
 For optional principal stress/strain output, together with the corresponding direction cosines, use Option 77.

Modeller See [Results Tables \(Appendix K\)](#).

Local Axes

Not applicable (global axes are the reference).

Sign Convention

- [Standard 3D continuum element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	For large displacements and large rotations.
Updated Lagrangian	For large displacements and large rotations.
Eulerian	For large displacements, large rotations and moderately large strains.
Co-rotational	For large displacements and large rotations.

Integration Schemes

Stiffness	Default.	1-point (TH4), 4-point (TH10), 3x2 (PN6, PN12, PN15), 2x2x2 (HX8, HX16, HX20)
	Fine (see <i>Options</i>).	5-point (TH10), 3x3x2 (HX16), 3x3x3 (HX20)
	Coarse (see <i>Options</i>)	13-point (HX20), 14-point (HX20)
Mass	Default.	1-point (TH4), 4-point (TH10), 3x2 (PN6, PN12, PN15), 2x2x2 (HX8, HX16, HX20)
	Fine (see <i>Options</i>).	4-point (TH4) 11-point (TH10), 14-point (TH10) 3x3x2 (HX16), 3x3x3 (HX20)
	Coarse (see <i>Options</i>)	13-point (HX20), 14-point (HX20)

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes fine integration rule.
- 36** Follower loads
- 54** Updated Lagrangian geometric nonlinearity
- 55** Output strains as well as stresses.
- 77** Output principal stresses and direction cosines.
- 87** Total Lagrangian geometric nonlinearity.
- 91** Invoke finer integration of the mass matrix.
- 102** Switch off load correction stiffness due to centripetal acceleration.
- 105** Lumped mass matrix.
- 139** Output yielded Gauss points only.

- 155 Use 14-point integration rule for HX20.
- 156 Use 13-point integration rule for HX20.
- 167 Eulerian geometric nonlinearity.
- 229 Co-rotational geometric nonlinearity.
- 395 Use 14-point integration rule for mass matrix of TH10 (used together with Option 91).
- 398 For HX20 and HX16 with fine integration use all integration points for stress extrapolation.

Notes on Use

1. The elements are based on the standard isoparametric approach. The variation of stresses within an element may be regarded as constant for the lower order elements (corner nodes only), and linear for the higher order elements (with mid-side nodes).
2. All elements pass the [patch test](#).
3. When using table input format for temperature dependent ORTHOTROPIC SOLID or ANISOTROPIC SOLID material properties, the value of nset used is that defined in the first line of the property table.
4. Non-conservative loading is available with these elements when using either Updated Lagrangian or Eulerian geometric nonlinear formulations together with the FLD loading facility.

Restrictions

- ☐ [Ensure mid-side node centrality](#)
- ☐ [Avoid excessive element curvature](#)
- ☐ Avoid excessive aspect ratio

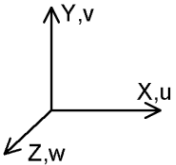
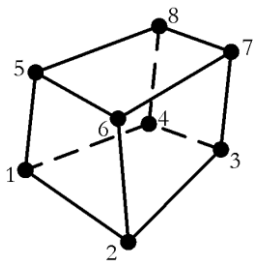
Recommendations on Use

- The 3D solid elements should be used if the stress field is fully 3D, i.e. it cannot be approximated with any of the 2D elements, e.g. as for a non-axisymmetric pressure vessel.
- For linear materials, the 20-noded element with a 2*2*2 Gauss rule is usually the most effective element, as this under-integration of the stiffness matrix prevents locking, i.e. over-stiff solutions will occur if the elements are used with a 3*3*3 Gauss integration rule to model structures subjected to bending. However, the element possesses six [zero energy modes](#). Therefore, a careful examination of the solution should be performed to check for spurious stress oscillations and peculiarities in the deformed configuration. Either the 14-point or 3*3*3 Gauss rules should be used for materially nonlinear problems or materially linear problems that exhibit spurious deformations.

- The 8-noded element should not be used for analyses where bending effects are significant as the element tends to lock in [parasitic shear](#) [C1]. The 8-noded element will perform poorly if it is highly distorted. The 4-noded tetrahedron TH4 element is generally not effective and should only be used if the geometry requires elements of this shape.

3D Solid Continuum Element with Enhanced Strains

General

Element Name	HX8M
	
Element Group	3D Continuum
Element Subgroup	Solid Continuum
Element Description	A 3D isoparametric solid element with an incompatible strain field. This mixed assumed strain element demonstrates a much superior performance to that of the HX8 element.
Number Of Nodes	8. The element is numbered according to a right-hand screw rule in the local z-direction.
Freedom Node	U, V, W: at each node.
Coordinates	X, Y, Z: at each node.

Geometric Properties

Not applicable.

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC SOLID (Elastic: Orthotropic Solid)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC SOLID (Elastic: Anisotropic Solid)
	Rigidities.	Not applicable.
Matrix	Not applicable.	
Joint	Not applicable.	
Concrete	MATERIAL PROPERTIES NONLINEAR 94 (Elastic: Isotropic, Plastic: Multi-Crack Concrete)	
	MATERIAL PROPERTIES NONLINEAR 102 (Elastic: Isotropic, Plastic: Smoothed Multi-Crack	

		Concrete)
Elasto-Plastic	Stress resultant:	Not applicable.
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Modified Mohr-Coulomb:	MATERIAL PROPERTIES MODIFIED MOHR_COULOMB (Elastic: Isotropic, Plastic: Mohr-Coulomb/Tresca, non-associative Hardening with tension/compression cut-off)
	Modified Cam-clay	MATERIAL PROPERTIES CAM_CLAY MODIFIED (Elastic: Isotropic, Plastic)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	MATERIAL PROPERTIES NONLINEAR 81 (Volumetric Crushing or Crushable Foam)
	Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
Creep	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic Shrinkage		VISCO ELASTIC PROPERTIES SHRINKAGE CEB_FIP_90, EUROCODE_2,

		GENERAL, USER
Rubber	Ogden:	MATERIAL PROPERTIES RUBBER OGDEN (Rubber: Ogden)
	Mooney-Rivlin:	MATERIAL PROPERTIES RUBBER MOONEY_RIVLIN (Rubber: Mooney-Rivlin)
	Neo-Hookean:	MATERIAL PROPERTIES RUBBER NEO_HOOKEAN (Rubber: Neo-Hookean)
	Hencky:	MATERIAL PROPERTIES RUBBER HENCKY (Rubber: Hencky)
Generic Polymer	Isotropic	MATERIAL PROPERTIES NONLINEAR 89 (Generic Polymer Model)
Composite	Not applicable.	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W: at each node.
Concentrated Loads	CL	Concentrated loads. Px, Py, Pz: at each node.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	Face Loads . Px, Py, Pz: local face pressures at nodes.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf, Ω_x , Ω_y , Ω_z , α_x , α_y , α_z
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, ϕ_4 , Xcbf, Ycbf, Zcbf
Velocities	VELO	Velocities. Vx, Vy, Vz: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay, Az: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. σ_x , σ_y , σ_z , σ_{xy} , σ_{yz} , σ_{xz} : global stresses. ϵ_x , ϵ_y , ϵ_z , γ_{xy} , γ_{yz} , γ_{xz} : global strains.
	SSIG	Initial stresses/strains at Gauss points σ_x , σ_y , σ_z , σ_{xy} , σ_{yz} , σ_{xz} : global stresses. ϵ_x , ϵ_y , ϵ_z , γ_{xy} , γ_{yz} , γ_{xz} : global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σ_x , σ_y , σ_z , σ_{xy} , σ_{yz} , σ_{xz} : global stresses.
	SSRG	Residual stresses at Gauss points. σ_x , σ_y , σ_z , σ_{xy} , σ_{yz} , σ_{xz} : global stresses.

Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver Stress (default): $\sigma_x, \sigma_y, \sigma_z, \sigma_{xy}, \sigma_{yz}, \sigma_{xz}, \sigma_e$: global stresses.
Strain: $\epsilon_x, \epsilon_y, \epsilon_z, \gamma_{xy}, \gamma_{yz}, \gamma_{xz}, \epsilon_e$: global strains.
Stretch (for rubber only): $V_{11}, V_{22}, V_{33}, V_{12}, V_{23}, V_{13}, \lambda_1, \lambda_2, \lambda_3, \det F$.
Where V_{ii} are components of the left stretch tensors, λ_i the principal stretches, $\theta\lambda$ the angle between the maximum principal stretch and the global X axis, and $\det F$ the determinant of the deformation gradient or volume ratio.
For optional principal stress/strain output, together with the corresponding direction cosines, use Option 77.

Modeller See [Results Tables \(Appendix K\)](#).

Local Axes

Not applicable (global axes are the reference).

Sign Convention

☐ [Standard 3D continuum element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	For large displacements and large rotations.
Updated Lagrangian	For large displacements and large rotations.
Eulerian	For large displacements, large rotations and moderately large strains.
Co-rotational	For large displacements and large rotations (large strains with the rubber material model).

Integration Schemes

Stiffness	Default.	2x2x2
	Fine.	As default.
Mass	Default.	2x2x2
	Fine.	As default.

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 39 Stress smoothing for rubber material models.
- 54 Updated Lagrangian geometric nonlinearity.
- 55 Output strains as well as stresses.
- 77 Output principal stresses and direction cosines.
- 87 Total Lagrangian geometric nonlinearity.
- 102 Switch off load correction stiffness due to centripetal acceleration.
- 105 Lumped mass matrix.
- 139 Output yielded Gauss points only.
- 167 Eulerian geometric nonlinearity.
- 225 Use alternative number of parameters for enhanced strain interpolation (see *Notes*).
- 229 Co-rotational geometric nonlinearity.

Notes on Use

1. The element is based on the standard isoparametric approach. The variation of stresses within an element may be regarded as linear.
2. The strain field for this element consists of two parts: the compatible strains derived from the assumed displacement field and the assumed enhanced strains; see *LUSAS Theory Manual*. By default, 18 parameters are used to define the assumed enhanced strain. In general, the default number of parameters should be used. However, 9 parameters may be specified using Option 225. In most cases the use of 9 or 18 parameters will give an equivalent solution. However, in some instances a better response may be obtained using more parameters at the expense of increased computation time.
3. The element passes the [patch test](#) and the large strain patch test for rubber.
4. When using table input format for temperature dependent ORTHOTROPIC SOLID or ANISOTROPIC SOLID material properties, the value of nset used is that defined in the first line of the property table.

5. Non-conservative (follower) loading is available with these elements when using either Updated Lagrangian or Eulerian geometric nonlinear formulations together with the FLD loading facility. The load does not have to be normal to the face and may also vary over the face.
6. To apply a non-conservative (follower) pressure load (load type FLD) with co-rotational geometric nonlinearity, Option 36 must be specified. Note that this load must be normal to the face and constant for all the nodes of the element face.
7. The converged stresses for the rubber material model are [Kirchhoff](#) stresses (see *LUSAS Theory Manual*).
8. Option 39 is used to smooth the stress output. It is particularly useful when the rubber material model is applied and the element is under very high compression where oscillatory stresses may appear (checker-board pattern).
9. For the rubber material model, converged values for strain output are replaced by the left stretch tensor V , the principal stretches of the vectors defining these principal directions. The principal stretches and directions can be obtained using Option 77. The value of $\det F = \lambda_1 \lambda_2 \lambda_3$ (the volume ratio) is only available for Gauss point output.
10. For the rubber material model, the iterative values of stress and strain are output in local co-rotated directions at the Gauss points only.
11. Convergence difficulties can sometimes arise when using enhanced strain elements with nonlinear materials, particularly if the material is elastic-perfectly plastic or if a very shallow hardening curve is defined. In such cases it is recommended that the standard element formulation is used.

Restrictions

- ☐ [Avoid excessive aspect ratio](#)
- ☐ Rubber material models can only be applied in conjunction with the co-rotational formulation, Option 229.

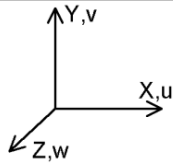
Recommendations on Use

This element exhibits an improved performance when compared with the parent element HX8. The integration rules are the same as the parent element. The HX8M element does not suffer from locking due to [parasitic shear](#) or when the material approaches the incompressible limit. No [zero energy modes](#) exist for this element.

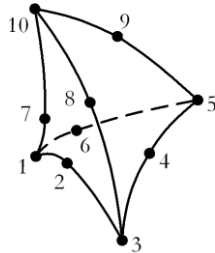
3D Solid Continuum Crack Tip Elements

General

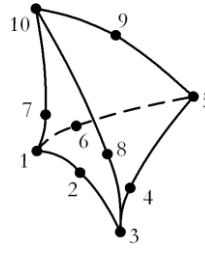
Element Name



TH10K

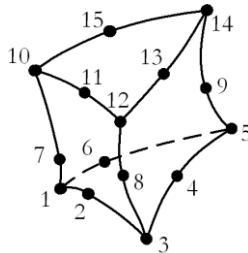


Crack specified at Node 1

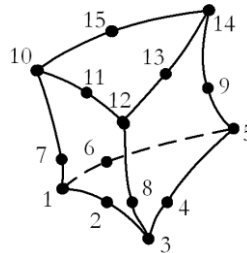


Crack specified along edge 1-2-3

PN15K

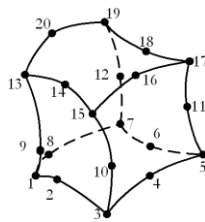


Crack specified at Node 1

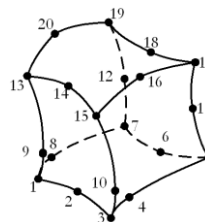


Crack specified along edge 1-2-3

HX20K



Crack specified at Node 1



Crack specified along edge 1-2-3

Element Group Element Subgroup Element Description

3D Continuum
Solid Continuum

A family of 3D isoparametric crack tip elements where the crack tip can be located at any corner node or along any edge of an element. The mid-side nodes are moved to the quarter points to produce a singularity at the crack tip. The strains vary as the square

	root of $1/R$, where R is the distance from the crack tip. These elements are used at the crack tip only. The elements are numerically integrated.
Number Of Nodes	10 (tetrahedra). 15 (pentahedra). 20 (hexahedra). The elements are numbered according to a right-hand screw rule in the local z -direction.
Freedoms	U, V, W: at each node.
Node Coordinates	X, Y, Z: at each node.

Geometric Properties

Not applicable.

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC SOLID (Elastic: Orthotropic Solid)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC SOLID (Elastic: Anisotropic Solid)
	Rigidities.	Not applicable.
Matrix	Not applicable.	
Joint	Not applicable.	
Concrete		MATERIAL PROPERTIES NONLINEAR 94 (Elastic: Isotropic, Plastic: Multi-Crack Concrete)
		MATERIAL PROPERTIES NONLINEAR 102 (Elastic: Isotropic, Plastic: Smoothed Multi-Crack Concrete)
Elasto-Plastic	Stress resultant:	Not applicable.
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Modified Mohr-Coulomb:	MATERIAL PROPERTIES MODIFIED MOHR_COULOMB (Elastic: Isotropic, Plastic: Mohr-Coulomb/Tresca, non-

		associative Hardening with tension/compression cut-off)
	Modified Cam-clay	MATERIAL PROPERTIES CAM_CLAY MODIFIED (Elastic: Isotropic, Plastic)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	MATERIAL PROPERTIES NONLINEAR 81 (Volumetric Crushing or Crushable Foam)
	Stress Potential:	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
Creep		CREEP PROPERTIES (Creep)
	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic		VISCO ELASTIC PROPERTIES
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Elasto-Plastic Interface		MATERIAL PROPERTIES NONLINEAR 26
Rubber	Not applicable.	
Generic Polymer	Isotropic	MATERIAL PROPERTIES NONLINEAR 89 (Generic Polymer Model)
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W: at each node.
Concentrated Loads	CL	Concentrated loads. Px, Py, Pz: at each node.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	Face Loads. Px, Py, Pz: local face pressures at nodes.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf, Ω_x , Ω_y , Ω_z , α_x , α_y , α_z
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, ϕ_4 , Xcbf, Ycbf, Zcbf
Velocities	VELO	Velocities. Vx, Vy, Vz: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay, Az: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. σ_x , σ_y , σ_z , σ_{xy} , σ_{yz} , σ_{xz} : global stresses. ϵ_x , ϵ_y , ϵ_z , γ_{xy} , γ_{yz} , γ_{xz} : global strains.
	SSIG	Initial stresses/strains at Gauss points σ_x , σ_y , σ_z , σ_{xy} , σ_{yz} , σ_{xz} : global stresses. ϵ_x , ϵ_y , ϵ_z , γ_{xy} , γ_{yz} , γ_{xz} : global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σ_x , σ_y , σ_z , σ_{xy} , σ_{yz} , σ_{xz} : global stresses.
	SSRG	Residual stresses at Gauss points. σ_x , σ_y , σ_z , σ_{xy} , σ_{yz} , σ_{xz} global stresses.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Stress (default): σ_x , σ_y , σ_z , σ_{xy} , σ_{yz} , σ_{xz} , σ_e : global stresses. Strain: ϵ_x , ϵ_y , ϵ_z , γ_{xy} , γ_{yz} , γ_{xz} , ϵ_e : global strains. For optional principal stress/strain output, together with the corresponding

direction cosines, use Option 77.

Modeller See [Results Tables \(Appendix K\)](#).

Local Axes

Not applicable (global axes are the reference).

Sign Convention

- ☐ [Standard 3D continuum element](#)

Formulation

Geometric Nonlinearity

- Total Lagrangian** For large displacements and large rotations.
- Updated Lagrangian** For large displacements and large rotations.
- Eulerian** For large displacements, large rotations and moderately large strains.
- Co-rotational** For large displacements and large rotations.

Integration Schemes

- Stiffness** Default. 4-point (TH10K), 6x3 (PN15K), 3x3x3 (HX20K)
Fine (see *Options*). 11-point (TH10K), 12x4 (HX15K)
- Mass** Default. 4-point (TH10K), 6x3 (PN15K), 3x3x3 (HX20K)
Fine (see *Options*). 11-point (TH10K), 14-point (TH10K), 12x4 (HX15K)

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes fine integration rule.
- 36** Follower loads
- 54** Updated Lagrangian geometric nonlinearity
- 55** Output strains as well as stresses.
- 77** Output principal stresses and direction cosines.
- 87** Total Lagrangian geometric nonlinearity.
- 91** Invoke finer integration of the mass matrix.

- 102** Switch off load correction stiffness due to centripetal acceleration.
- 105** Lumped mass matrix.
- 139** Output yielded Gauss points only.
- 167** Eulerian geometric nonlinearity.
- 229** Co-rotational geometric nonlinearity.
- 395** Use 14-point integration rule for mass matrix of TH10 (used together with Option 91).
- 398** For HX20 and HX16 with fine integration use all integration points for stress extrapolation.

Notes on Use

1. The elements are based on the standard isoparametric approach. Moving the mid-side nodes to the quarter points creates a singularity with theoretically infinite stress at the crack tip.
2. When using table input format for temperature dependent ORTHOTROPIC SOLID or ANISOTROPIC SOLID material properties, the value of nset used is that defined in the first line of the property table.
3. Non-conservative loading is available with these elements when using either Updated Lagrangian or Eulerian geometric nonlinear formulations together with the FLD loading facility.

Restrictions

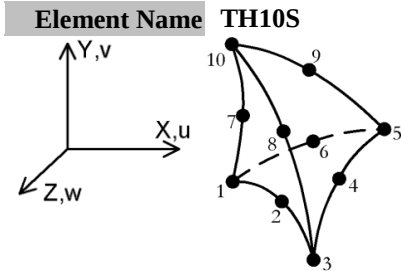
- ☐ [Avoid excessive element curvature](#)
- ☐ Avoid excessive aspect ratio

Recommendations on Use

- The 3D solid crack tip elements should be used if the stress field is fully 3D, i.e. it cannot be approximated with any of the 2D crack tip elements.
- Elements TH10K, PN15K and HX20K are specifically designed for application to fracture mechanics problems and may be used to model the singularities that occur at the crack tip. The mid-side nodes near the crack tip are shifted to the quarter point. This ensures a singularity is present at the crack tip and that strains vary as $1/\sqrt{r}$ where r is the distance from the crack tip.

3D Solid Continuum Composite Elements (Tetrahedral)

General



Element Name	TH10S
Element Group	3D Continuum
Element Subgroup	Solid Continuum
Element Description	A 3D tetrahedral element capable of modelling curved boundaries. The element can be arbitrarily oriented with respect to the laminate and allows for the fully automatic mesh generation of laminate geometric models imported from CAD packages.
Number Of Nodes	10. The element is numbered according to a right-hand screw rule in the local z-direction.
Freedom	U, V, W: at each node.
Node Coordinates	X, Y, Z: at each node.

Geometric Properties

See [Composites](#) in the *Modeller Reference Manual*

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC SOLID (Elastic: Orthotropic Solid)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC SOLID (Elastic: Anisotropic Solid)
	Rigidities.	Not applicable.
Matrix	Not applicable.	
Joint	Not applicable.	
Concrete		MATERIAL PROPERTIES NONLINEAR 94 (Elastic: Isotropic, Plastic: Multi-Crack Concrete)
		MATERIAL PROPERTIES NONLINEAR 102 (Elastic: Isotropic, Plastic: Smoothed Multi-Crack)

		Concrete)
Elasto-Plastic	Stress resultant:	Not applicable.
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Modified Mohr-Coulomb:	MATERIAL PROPERTIES MODIFIED MOHR_COULOMB (Elastic: Isotropic, Plastic: Mohr-Coulomb/Tresca, non-associative Hardening with tension/compression cut-off)
	Volumetric Crushing:	Not applicable.
	Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
Creep	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic Shrinkage		VISCO ELASTIC PROPERTIES SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber Generic Polymer	Not applicable.	MATERIAL PROPERTIES NONLINEAR 89 (Generic Polymer Model)

Resin Cure Model		MATERIAL PROPERTIES NONLINEAR CURE LAYER, FIBRE_RESIN
Composite	Composite solid:	COMPOSITE PROPERTIES (Elastic: Orthotropic Solid)

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W: at each node.
Concentrated Loads	CL	Concentrated loads. Px, Py, Pz: at each node.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	Face Loads . Px, Py, Pz: local face pressures at nodes.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf, Ω_x , Ω_y , Ω_z , α_x , α_y , α_z
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, ϕ_4 , Xcbf, Ycbf, Zcbf
Velocities	VELO	Velocities. Vx, Vy, Vz: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay, Az: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. σ_x , σ_y , σ_z , σ_{xy} , σ_{yz} , σ_{xz} : global stresses. ϵ_x , ϵ_y , ϵ_z , γ_{xy} , γ_{yz} , γ_{xz} : global strains.
	SSIG	Initial stresses/strains at Gauss points (see <i>Notes</i>). σ_x , σ_y , σ_z , σ_{xy} , σ_{yz} , σ_{xz} : global stresses. ϵ_x , ϵ_y , ϵ_z , γ_{xy} , γ_{yz} , γ_{xz} : global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σ_x , σ_y , σ_z , σ_{xy} , σ_{yz} , σ_{xz} : global stresses.
	SSRG	Residual stresses at Gauss points (see <i>Notes</i>). σ_x , σ_y , σ_z , σ_{xy} , σ_{yz} , σ_{xz} : global stresses.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Stress (default): σ_x , σ_y , σ_z , σ_{xy} , σ_{yz} , σ_{xz} : local stresses.

Strain: $\epsilon_x, \epsilon_y, \epsilon_z, \gamma_{xy}, \gamma_{yz}, \gamma_{xz}$: local strains.

Stresses and strains are output at the Gauss and corner points of the subdivision(s) of each layer. For optional principal stress/strain output, together with the corresponding direction cosines, use Option 77.

Modeller See [Results Tables \(Appendix K\)](#).

Local Axes

The local axes for each layer are defined by the LAMINAR DIRECTIONS specified for its bottom surface. The three node set in LAMINAR DIRECTIONS define the local Cartesian set origin, the x-axis and the positive quadrant of the xy-plane respectively. The local z-axis forms an orthonormal coordinate system with x and y.

Sign Convention

- ☐ [Standard 3D continuum element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	Not applicable.
Updated Lagrangian	Not applicable.
Eulerian	Not applicable.
Co-rotational	For large displacements and large rotations.

Integration Schemes

Stiffness	Default.	1-point for a tetrahedral subdivision (see Notes), 3-point for a pentahedral/pyramid subdivision, 2x2 for a hexahedral/wrick subdivision
	Fine (see <i>Options</i>).	1-point for a tetrahedral subdivision (see Notes), 3x2 for a pentahedral/pyramid subdivision, 2x2 x2 for a hexahedral/wrick subdivision
Mass	Default	5-point for the whole element or (see Options) 1-point for a tetrahedral subdivision, 3x2 for a pentahedral/pyramid subdivision, 2x2 x2 for a hexahedral/wrick subdivision
	Fine (see <i>Options</i>).	11-point or (see Options) 14 -point for the whole element

Mass Modelling

- ☐ Consistent mass (default).

- ☐ Lumped mass.

Options

- 18** Invokes fine integration rule.
- 36** Follower loads.
- 55** Output strains as well as stresses.
- 77** Output principal stresses and direction cosines.
- 91** Formulate element mass with fine integration.
- 105** Lumped mass matrix.
- 139** Output yielded Gauss points only.
- 229** Co-rotational geometric nonlinearity.
- 266** Layer by layer computation of mass matrix.
- 394** Lamina directions supported.
- 395** Use 14-point fine integration rule for mass matrix of TH10 family (used together with 91).

Notes on Use

1. The element is based on the standard isoparametric approach. The variation of strains within an element may be regarded as linear.
2. All elements pass the [patch test](#).
3. The LAMINAR DIRECTIONS and COMPOSITE PROPERTIES data chapters must be used with this element in conjunction with the COMPOSITE ASSIGNMENTS data chapter.
4. The stresses obtained from a geometric nonlinear analysis are [Kirchhoff](#) stresses.
5. If the whole tetrahedral element is embedded in a single lamina, a 4-point integration rule will be used for this tetrahedral subdivision; otherwise a 1-point rule will be used.
6. The mass matrix can be computed using a layer by layer integration (OPTION 266), however this should only be used when the densities of the layers vary considerably because the computation time can be greatly increased when this OPTION is specified.
7. Numerical integration through the thickness is performed. The integration points are located in the subdivisions of each layer. Each subdivision forms the shape of a regular 3D solid continuum element and the integration points are located accordingly within the subdivision as described above.
8. SSIG and SSRG loads have to be applied at the Gauss point positions for the subdivision(s) of each layer.
9. Layer 1 is always the bottom layer.

Restrictions

- ☐ Ensure mid-side node centrality
- ☐ Avoid excessive element curvature
- ☐ Avoid excessive aspect ratio

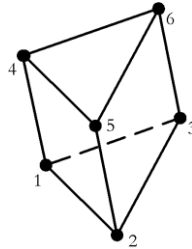
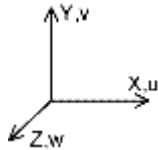
Recommendations on Use

- 3D solid composite elements should be used for modelling thick composite structures comprising laminae of differing material properties where the computational cost of modelling each lamina with an individual solid element would be prohibitive.
- As these elements can be arbitrarily oriented with respect to the laminate, they are particularly aimed at the use of fully automatic mesh generation of laminate geometric models imported from CAD packages.

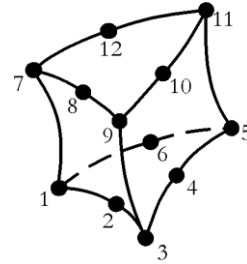
3D Solid Continuum Composite Elements (Pentahedral and Hexahedral)

General

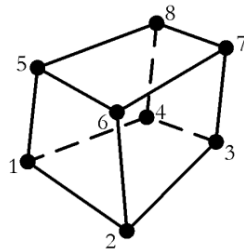
Element Name PN6L



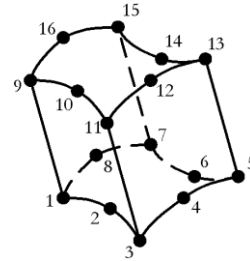
PN12L



HX8L



HX16L



Element Group
Element Subgroup
Element Description

3D Continuum
Solid Continuum

Number Of Nodes

3D isoparametric pentahedral and hexahedral solid elements with higher order models capable of modelling curved boundaries. The element can be used to model a laminate, where lamina planes are defined by the top and bottom surfaces of the element. The elements are numerically integrated.

Node Coordinates

U, V, W: at each node.
X, Y, Z: at each node.

Geometric Properties

See [Composites](#) in the *Modeller Reference Manual*

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC SOLID (Elastic: Orthotropic Solid)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC SOLID (Elastic: Anisotropic Solid)
	Rigidities.	Not applicable.
Matrix	Not applicable.	
Joint	Not applicable.	
Concrete		MATERIAL PROPERTIES NONLINEAR 94 (Elastic: Isotropic, Plastic: Multi-Crack Concrete)
		MATERIAL PROPERTIES NONLINEAR 102 (Elastic: Isotropic, Plastic: Smoothed Multi-Crack Concrete)
Elasto-Plastic	Stress resultant:	Not applicable.
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Modified Mohr-Coulomb:	MATERIAL PROPERTIES MODIFIED MOHR_COULOMB (Elastic: Isotropic, Plastic: Mohr-Coulomb/Tresca, non-associative Hardening with tension/compression cut-off)
	Volumetric Crushing:	Not applicable.
	Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
	Creep CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE

		(Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE
		(Concrete creep model to EUROCODE_2)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic		VISCO ELASTIC PROPERTIES
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable.	
Generic		MATERIAL PROPERTIES NONLINEAR 89
Polymer		(Generic Polymer Model)
Resin Cure		MATERIAL PROPERTIES NONLINEAR CURE
Model		LAYER, FIBRE_RESIN
Composite	Composite solid:	COMPOSITE PROPERTIES (Elastic: Orthotropic Solid)

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W: at each node.
Concentrated Loads	CL	Concentrated loads. Px, Py, Pz: at each node.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	Face Loads . Px, Py, Pz: local face pressures at nodes.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf, Ω_x , Ω_y , Ω_z , α_x , α_y , α_z
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, ϕ_4 , Xcbf, Ycbf, Zcbf
Velocities	VELO	Velocities. Vx, Vy, Vz: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay, Az: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. σ_x , σ_y , σ_z , σ_{xy} , σ_{yz} , σ_{xz} : global stresses. ϵ_x , ϵ_y , ϵ_z , γ_{xy} , γ_{yz} , γ_{xz} : global strains.
	SSIG	Initial stresses/strains at Gauss points (see <i>Notes</i>). σ_x , σ_y , σ_z , σ_{xy} , σ_{yz} , σ_{xz} : global stresses. ϵ_x , ϵ_y , ϵ_z , γ_{xy} , γ_{yz} , γ_{xz} : global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element.

	SSRG	$\sigma_x, \sigma_y, \sigma_z, \sigma_{xy}, \sigma_{yz}, \sigma_{xz}$: global stresses. Residual stresses at Gauss points (see <i>Notes</i>).
Temperatures	TEMP, TMPE	$\sigma_x, \sigma_y, \sigma_z, \sigma_{xy}, \sigma_{yz}, \sigma_{xz}$ global stresses. Temperatures at nodes/for element. $T, 0, 0, 0, T_o, 0, 0, 0$
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver Stress (default): $\sigma_x, \sigma_y, \sigma_z, \sigma_{xy}, \sigma_{yz}, \sigma_{xz}$: local stresses.

Strain: $\epsilon_x, \epsilon_y, \epsilon_z, \gamma_{xy}, \gamma_{yz}, \gamma_{xz}$: local strains.

Stresses and strains are output at the top and bottom of each layer. For optional principal stress/strain output, together with the corresponding direction cosines, use Option 77.

Modeller See [Results Tables \(Appendix K\)](#).

Local Axes

The local axes for each layer are defined using the convention for [standard area elements](#). Local axes are computed at the top and bottom surfaces (at the Gauss points) and average values are interpolated for the mid-surface. The top and bottom faces of the element are as shown, e.g. nodes 1, 2, 3, 4 define the bottom face of HX8L. Every layer uses the same averaged values.

Sign Convention

☐ [Standard 3D continuum element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	Not applicable.
Updated Lagrangian	Not applicable.
Eulerian	Not applicable.
Co-rotational	For large displacements and large rotations.

Integration Schemes

Stiffness Default. 1-point for each layer (PN6L), 3-point for each layer (PN12L),

		2x2 for each layer (HX8L,HX16L)
	Fine (see <i>Options</i>).	3-point for each layer (PN6L), 3x3 for each layer (HX16L)
Mass	Default	3x2 for the whole element (PN6L,PN12L) or (see Options) 1-point for each layer (PN6L), 3-point for each layer (PN12L),2x2x2 for the whole element or 2x2 for each layer (HX8L,HX16L)
	Fine (see <i>Options</i>).	3x2 for the whole element or 3-point for each layer (PN6L),3x3x2 for the whole element or 3x3 for each layer (HX16L)

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes fine integration rule.
- 36** Follower loads.
- 55** Output strains as well as stresses.
- 77** Output principal stresses and direction cosines.
- 105** Lumped mass matrix.
- 139** Output yielded Gauss points only.
- 229** Co-rotational geometric nonlinearity.
- 266** Layer by layer computation of mass matrix.
- 303** Exclude incompatible modes for solid composite elements.

Notes on Use

1. The elements are based on the standard isoparametric approach. The variation of stresses within an element may be regarded as constant for the lower order elements (corner nodes only), and linear in the plane of the quadratic element faces for the higher order elements.
2. All elements pass the [patch test](#).
3. The COMPOSITE GEOMETRY and COMPOSITE PROPERTIES data chapters must be used with this element in conjunction with the COMPOSITE ASSIGNMENTS data chapter.
4. The stresses obtained from a geometric nonlinear analysis are [Kirchhoff](#) stresses.
5. The mass matrix can be computed using a layer by layer integration (Option 266), however this should only be used when the densities of the layers vary

considerably because the computation time can be greatly increased applying this option.

6. Numerical integration through the thickness is performed. The integration points are located at the top and bottom surface of each layer.
7. SSIG and SSRG loads have to be applied at the Gauss point positions for the top and bottom surfaces of each layer.
8. Layer 1 is always the bottom layer.

Restrictions

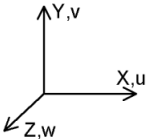
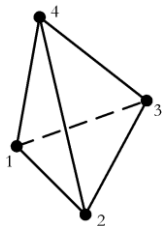
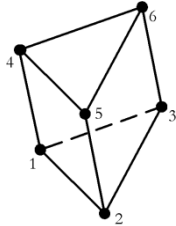
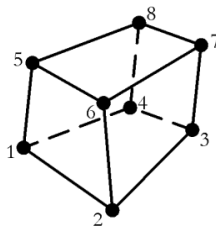
- ☐ [Ensure mid-side node centrality](#)
- ☐ [Avoid excessive element curvature](#)
- ☐ Avoid excessive aspect ratio

Recommendations on Use

- The 3D solid composite elements should be used for modelling thick composite structures comprising laminae of differing material properties where the computational cost of modelling each lamina with an individual solid element would be prohibitive.
- Because of the numerical integration through the thickness, by increasing the number of layers the accuracy of solution will increase. This can be achieved by dividing each single layer into two or three identical layers.

3D Solid Continuum Explicit Dynamics Elements

General

Element Name	TH4E	PN6E	HX8E
			
Element Group	3D Continuum		
Element Subgroup	Solid Continuum		
Element Description	A family of 3D isoparametric solid elements for explicit dynamic analyses. The elements are numerically integrated.		
Number Of Nodes	4 (tetrahedra), 6 (pentahedra), 8 (hexahedra). The elements are numbered according to a right-hand screw rule in the local z-direction.		
Freedom	U, V, W: at each node.		
Node Coordinates	X, Y, Z: at each node.		

Geometric Properties

Not applicable.

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC SOLID (Elastic: Orthotropic Solid)
	Anisotropic:	Not applicable.
	Rigidities.	Not applicable.
Matrix	Not applicable	
Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic	Stress resultant:	Not applicable.
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic)

		Strain or Isotropic Total Strain)
Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64	(Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65	(Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
Modified Mohr-Coulomb:	MATERIAL PROPERTIES MODIFIED MOHR_COULOMB	(Elastic: Isotropic, Plastic: Mohr-Coulomb/Tresca, non-associative Hardening with tension/compression cut-off)
Modified Cam-clay	MATERIAL PROPERTIES CAM_CLAY MODIFIED	(Elastic: Isotropic, Plastic)
Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75	(Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
Volumetric Crushing:	MATERIAL PROPERTIES NONLINEAR 81	(Volumetric Crushing or Crushable Foam)
Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN	(Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
Creep		CREEP PROPERTIES (Creep) (see <i>Notes</i>)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic		VISCO ELASTIC PROPERTIES
Shrinkage	Not applicable	
Rubber	Not applicable	
Generic Polymer	Not applicable	
Composite	Not applicable	
Loading		
Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W: at each node.
Concentrated Loads	CL	Concentrated loads. Px, Py, Pz: at each node.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	Face Loads . Px, Py, Pz: local face pressures at

		nodes.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf, Ω_x , Ω_y , Ω_z , α_x , α_y , α_z
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, ϕ_4 , Xcbf, Ycbf, Zcbf
Velocities	VELO	Velocities. Vx, Vy, Vz: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay, Az: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. σ_x , σ_y , σ_z , σ_{xy} , σ_{yz} , σ_{xz} : global stresses. ϵ_x , ϵ_y , ϵ_z , γ_{xy} , γ_{yz} , γ_{xz} : global strains.
	SSIG	Not applicable.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σ_x , σ_y , σ_z , σ_{xy} , σ_{yz} , σ_{xz} : global stresses.
	SSRG	Residual stresses at Gauss points. σ_x , σ_y , σ_z , σ_{xy} , σ_{yz} , σ_{xz} : global stresses.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Stress(default): σ_x , σ_y , σ_z , σ_{xy} , σ_{yz} , σ_{xz} , σ_e : global stresses. Strain: not available (see <i>Notes</i>). For optional principal stress output, together with the corresponding direction cosines, use Option 77.
Modeller	See Results Tables (Appendix K) .

Local Axes

Not applicable (global axes are the reference).

Sign Convention

- [Standard 3D continuum element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	Not applicable.
Updated Lagrangian	Not applicable.
Eulerian	For large displacements, large rotations and moderately large strains.
Co-rotational	For large displacements and large rotations.

Integration Schemes

Stiffness	Default.	1-point (see <i>Notes</i>).
	Fine.	As default.
Mass	Default.	1-point (see <i>Notes</i>).
	Fine.	As default.

Mass Modelling

- ☐ Lumped mass only (see *Notes*).

Options

- 77 Output principal stresses and direction cosines.
105 Lumped mass matrix.
139 Output yielded Gauss points only.

Notes on Use

1. The elements are based on the standard isoparametric approach. Stresses within an element may be regarded as constant.
2. When using tabular input for ORTHOTROPIC SOLID the value of nset used is that defined in the first line of the property table.
3. The system parameter HGVISC is used to restrict element mechanisms due to under-integration. The default value is usually sufficient.
4. The bulk viscosity coefficients are used to restrict numerical oscillations due to the traversal of stress waves. The default bulk viscosity coefficients (BULKLF and BULKQF) may be altered as SYSTEM parameters.
5. These elements must be used with a dynamic central difference scheme and a lumped mass matrix.
6. These element are Not applicable. for static or eigenvalue analyses.

7. Automatic time step length calculations are implemented.
8. As element geometry is always updated in an explicit dynamic analysis, the solution is nonlinear. When using explicit dynamic elements **NONLINEAR CONTROL** must be specified.
9. If **CREEP PROPERTIES** are defined, explicit time integration must be specified in **VISCOUS CONTROL**.
10. Strains are computed incrementally and therefore total strains are not available for output.
11. Non-conservative loading is invoked when the **FLD** loading facility is applied.
12. Rayleigh damping coefficients are not supported by these elements.
13. Constraint equations are not available for use with these elements.

Restrictions

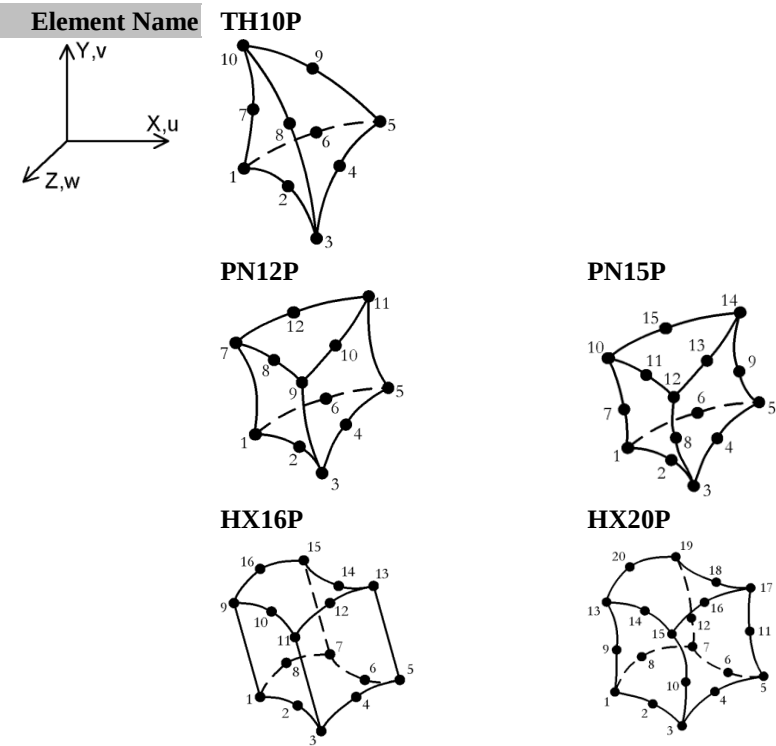
- ❑ [Avoid excessive aspect ratio](#)

Recommendations on Use

- Explicit dynamics elements may be used to define surface boundaries which will be active in a slideline analysis.
- The 3D explicit dynamics elements should be used if the stress field is fully 3D, i.e. it cannot be approximated with any of the 2D elements, e.g. a non-axisymmetric pressure vessel.

3D Solid Two Phase Continuum Elements

General



Element Group	3D Continuum
Element Subgroup	Solid Continuum
Element Description	A family of 3D isoparametric solid two phase continuum elements capable of modelling curved boundaries. The elements are numerically integrated.
Number Of Nodes	10 (tetrahedra). 12 or 15 (pentahedra). 16 or 20 (hexahedra). The elements are numbered according to a right-hand screw rule in the local z-direction.
Freedoms	U, V, W, P: at corner nodes, U, V, W at mid-side nodes.
Node Coordinates	X, Y, Z: at each node.

Geometric Properties

Not applicable.

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC SOLID (Elastic: Orthotropic Solid)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC SOLID (Elastic: Anisotropic Solid)
	Rigidities.	Not applicable.
Matrix	Not applicable.	
Joint	Not applicable.	
Concrete		MATERIAL PROPERTIES NONLINEAR 94 (Elastic: Isotropic, Plastic: Multi-Crack Concrete)
		MATERIAL PROPERTIES NONLINEAR 102 (Elastic: Isotropic, Plastic: Smoothed Multi-Crack Concrete)
Elasto-Plastic	Stress resultant:	Not applicable.
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Modified Mohr-Coulomb:	MATERIAL PROPERTIES MODIFIED MOHR_COULOMB (Elastic: Isotropic, Plastic: Mohr-Coulomb/Tresca, non-associative Hardening with tension/compression cut-off)
	Modified Cam-clay	MATERIAL PROPERTIES CAM_CLAY MODIFIED (Elastic: Isotropic, Plastic)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	MATERIAL PROPERTIES NONLINEAR 81 (Volumetric Crushing or Crushable Foam)
	Stress Potential:	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
	Creep	CREEP PROPERTIES (Creep)

Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic		VISCO ELASTIC PROPERTIES
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Elasto-Plastic Interface		MATERIAL PROPERTIES NONLINEAR 26
Rubber	Not applicable.	
Generic Polymer	Isotropic	MATERIAL PROPERTIES NONLINEAR 89 (Generic Polymer Model)
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W, P: at corner nodes, U, V, W at mid-side nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py, Pz, Q: at corner nodes, .Px, Py, Pz at mid-side nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	Face Loads . Px, Py, Pz, Q: face pressures/flux per unit area at corner nodes relative to local face axes. Px, Py, Pz: face pressures at midside nodes relative to local face axes.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf, Ω_x , Ω_y , Ω_z , α_x , α_y , α_z , gx, gy, gz. (See notes on use)
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, ϕ_4 , Xcbf, Ycbf, Zcbf, gx, gy, gz. (See notes on use)
Velocities	VELO	Velocities. Vx, Vy, Vz: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay, Az: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. σ_x , σ_y , σ_z , σ_{xy} , σ_{yz} , σ_{xz} , σ_p global stresses. ϵ_x , ϵ_y , ϵ_z , γ_{xy} , γ_{yz} , γ_{xz} : global strains.
	SSIG	Initial stresses/strains at Gauss points σ_x , σ_y ,

		$\sigma_z, \sigma_{xy}, \sigma_{yz}, \sigma_{xz}, \sigma_p$: global stresses. $\epsilon_x, \epsilon_y, \epsilon_z, \gamma_{xy}, \gamma_{yz}, \gamma_{xz}$: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. $\sigma_x, \sigma_y, \sigma_z, \sigma_{xy}, \sigma_{yz}, \sigma_{xz}, \sigma_p$: global stresses.
	SSRG	Residual stresses at Gauss points. $\sigma_x, \sigma_y, \sigma_z, \sigma_{xy}, \sigma_{yz}, \sigma_{xz}, \sigma_p$ global stresses.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Stress (default): $\sigma_x, \sigma_y, \sigma_z, \sigma_{xy}, \sigma_{yz}, \sigma_{xz}, \sigma_p, \sigma_e$: global stresses. Strain: $\epsilon_x, \epsilon_y, \epsilon_z, \gamma_{xy}, \gamma_{yz}, \gamma_{xz}, \epsilon_v, \epsilon_e$: global strains. For optional principal stress/strain output, together with the corresponding direction cosines, use Option 77.
Modeller	See Results Tables (Appendix K) .

Local Axes

Not applicable (global axes are the reference).

Sign Convention

- ☐ [Standard 3D continuum element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	For large displacements and large rotations.
Updated Lagrangian	For large displacements and large rotations.
Eulerian	For large displacements, large rotations and moderately large strains.
Co-rotational	For large displacements and large rotations.

Integration Schemes

Stiffness	Default.	4-point (TH10P), 3x2 (PN12P, PN15P), 2x2x2 (HX16P, HX20P)
	Fine (see <i>Options</i>).	5-point (TH10P), 3x3x2 (HX16P), 3x3x3 (HX20P)
	Coarse (see <i>Options</i>)	13-point (HX20P), 14-point (HX20P)
Mass	Default.	4-point (TH10P), 3x2 (PN12P, PN15P), 2x2x2 (HX16P, HX20P)
	Fine (see <i>Options</i>).	11-point (TH10P), 14-point (TH10P), 3x3x2 (HX16P), 3x3x3 (HX20P)
	Coarse (see <i>Options</i>)	13-point (HX20P), 14-point (HX20P)

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes fine integration rule.
- 36** Follower loads
- 54** Updated Lagrangian geometric nonlinearity
- 55** Output strains as well as stresses.
- 77** Output principal stresses and direction cosines.
- 87** Total Lagrangian geometric nonlinearity.
- 91** Invoke finer integration of the mass matrix.
- 102** Switch off load correction stiffness due to centripetal acceleration.
- 105** Lumped mass matrix.
- 139** Output yielded Gauss points only.
- 155** Use 14-point integration rule for HX20P.
- 156** Use 13-point integration rule for HX20P.
- 167** Eulerian geometric nonlinearity.
- 229** Co-rotational geometric nonlinearity.
- 398** For HX20P and HX16P with fine integration use all integration points for stress extrapolation.

Notes on Use

1. Two phase material parameters must be used with these elements for undrained and consolidation analysis.

2. The elements are based on the standard isoparametric approach. The variation of stresses and pore pressures within an element may be regarded linear, except for elements PN12P and HX16P where the stress is constant in the z direction.
3. All elements pass the [patch test](#).
4. When using table input format for temperature dependent ORTHOTROPIC SOLID or ANISOTROPIC SOLID material properties, the value of nset used is that defined in the first line of the property table.
5. Non-conservative loading is available with these elements when using Updated Lagrangian, Eulerian or co-rotational (with OPTION 36) geometric nonlinear formulations together with the FLD loading facility.
6. The global components of gravity acting on the fluid phase are defined by gx and gy under CBF and BF loading

Restrictions

- ☐ [Ensure mid-side node centrality](#)
- ☐ [Avoid excessive element curvature](#)
- ☐ Avoid excessive aspect ratio

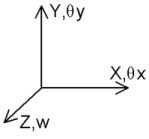
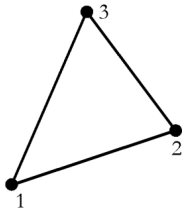
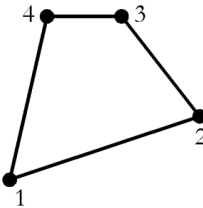
Recommendations on Use

- The 3D solid two phase elements should be used if the stress field is fully 3D, i.e. it cannot be approximated with any of the 2D elements, e.g. a non-axisymmetric pressure vessel.
- For linear materials, the 20-noded element with a 2*2*2 Gauss rule is usually the most effective element, as this under-integration of the stiffness matrix prevents locking, i.e. over-stiff solutions will occur if the elements are used with a 3*3*3 Gauss integration rule to model structures subjected to bending. However, the element possesses six [zero energy modes](#). Therefore, a careful examination of the solution should be performed to check for spurious stress oscillations and peculiarities in the deformed configuration. Either the 14-point or 3*3*3 Gauss rules should be used for materially nonlinear problems or materially linear problems that exhibit spurious deformations.
- In general, PN15P and HX20P give the best performance; TH10P is less accurate and needs to be used with a finer mesh. HX16P and PN12P should only be used to overcome connectivity problems when meshing.

Chapter 5 : Plate Elements.

2D Isoflex Thin Plate Flexure Elements

General

Element Name	TF3	QF4
		
Element Group	Plates	
Element Subgroup	Isoflex Plates	
Element Description	A family of thin plate flexure elements in 2D with higher order models capable of modelling curved boundaries. The element formulation takes account of varying thickness and anisotropic properties. As required by thin plate theory, transverse shearing effects are excluded.	
Number Of Nodes	3 or 4 numbered anticlockwise.	
Freedom	w, θ_x, θ_y : at the corner nodes.	
Node Coordinates	X, Y : at each node.	

Geometric Properties

$t_1 \dots t_n$ Thickness at each node.

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC (Elastic: Orthotropic Plane Stress)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC 3 (Elastic: Anisotropic Thin Plate)
	Rigidities.	RIGIDITIES 3 (Rigidities: Membrane/Thin Plate)
Matrix	Not applicable	
Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic	Not applicable	

Creep	Not applicable
Damage	Not applicable
Viscoelastic	Not applicable
Shrinkage	Not applicable
Rubber	Not applicable
Generic	Not applicable
Polymer	
Composite	Not applicable

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. W , θ_x , θ_y : at the corner nodes.
Concentrated Loads	CL	Concentrated loads. P_z , M_x , M_y : at corner nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Uniformly distributed loads. W_z : normal pressure for element (global).
	FLD	Not applicable.
Body Forces	CBF	Constant body forces for element. Z_{cbf}
	BFP, BFPE	Body force potentials at nodes/for element. ϕ_1 , Z_{cbf}
Velocities	VELO	Velocities. V_z : at nodes.
Accelerations	ACCE	Accelerations. A_z : at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. M_x , M_y , M_{xy} : moments/unit width (global). ψ_x , ψ_y , ψ_{xy} : flexural strains (global).
	SSIG	Not applicable.
Residual Stresses	Not applicable.	
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. 0 , 0 , 0 , dT/dz , 0 , 0 , 0 , dTo/dz
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Stress resultant: M_x , M_y , M_{xy} : moments/unit width (global). Strain: ψ_x , ψ_y , ψ_{xy} : flexural strains (global).
Modeller	See Results Tables (Appendix K) .

Local Axes

Not applicable (global axes are the reference).

Sign Convention

- ☐ [Standard plate element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Stiffness	Default.	3-point (TF3), 2x2 (QF4).
	Fine.	As default.
Mass	Default.	3-point (TF3), 2x2 (QF4).
	Fine.	As default.

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes fine integration rule for element.
- 55** Output strains as well as stresses.
- 143** Output shear forces for low order thin plate bending elements.
- 170** Suppress transfer of shape function arrays to disk.

Notes on Use

1. The element formulations are based on an [Kirchhoff](#) hypothesis for thin plates.
2. The variation of moments within the elements can be regarded as linear.
3. The elements pass the [patch test](#) for convergence for mixed triangular and quadrilateral element geometry.
4. The averaged nodal values produced with ELEMENT OUTPUT do not include the thin [isoflex](#) plate shear stresses if Option 143 is invoked.
5. When Option 143 is invoked shear stresses are only computed for the low order isoflex elements (QF4,TF3).

Restrictions

- ❑ [Avoid excessive aspect ratio](#)

Recommendations on Use

- This element may be used to analyse any 2D plate type structures where transverse shear effects do not influence the solution, e.g. thin cantilever plates.
- The thick plate elements QTF8 and TTF6 are recommended for thick plates where transverse shear strains are no longer negligible.

The following element combinations should be used for ribbed plates;

Ribs with small or no eccentricity

- ❑ QSI4/TS3 elements with BMS3 elements,
- ❑ QTS4/TTS3 elements with BMS3 elements.

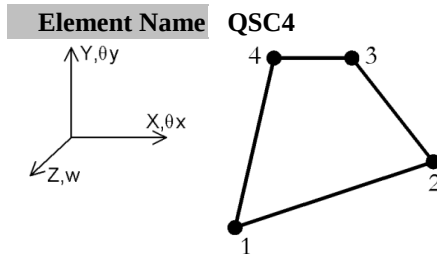
Ribs with large eccentricity

- ❑ QSL8/TSL6 elements with BSL3/BSL4/BXL4 elements.
- ❑ QTS4/TTS3 elements with BTS3 elements.

The through thickness integration is performed explicitly.

2D Isoflex Thick Plate Flexure Element

General



Element Group	Plates
Element Subgroup	Isoflex Plates
Element Description	A thick plate flexure element in 2D. The element formulation takes into account varying thickness and anisotropic properties. Transverse shearing effects are included.
Number Of Nodes	4, numbered anticlockwise.
Freedom	w, θ_x, θ_y : at each node.
Node Coordinates	X, Y : at each node.

Geometric Properties

$t_1 \dots t_n$ At each node.

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC THICK (Elastic: Orthotropic Thick)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC 5 (Elastic: Anisotropic Thick Plate)
	Rigidities:	RIGIDITIES 5 (Rigidities: Thick Plate)
Matrix	Not applicabl	
Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic	Not applicable	
Creep	Not applicable.	
Damage	Not applicable	

Viscoelastic	Not applicable
Shrinkage	Not applicable
Rubber	Not applicable
Generic	Not applicable
Polymer	
Composite	Not applicable

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. W , θ_x , θ_y : at nodes.
Concentrated Loads	CL	Concentrated loads. P_z , M_x , M_y : at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Uniformly distributed loads. W_z : normal pressure for element (global).
	FLD	Not applicable.
Body Forces	CBF	Constant body forces for element. Z_{cbf}
	BFP, BFPE	Body force potentials at nodes/for element. ϕ_1 , Z_{cbf}
Velocities	VELO	Velocities. V_z : at nodes.
Accelerations	ACCE	Accelerations. A_z : at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. M_x , M_y , M_{xy} : moments/unit width (global).
		ψ_x , ψ_y , ψ_{xy} : flexural strains (global).
	SSIG	Not applicable.
Residual Stresses	Not applicable.	
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. 0, 0, 0, dT/dz , 0, 0, 0, dT_0/dz
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Stress resultant: M_x , M_y , M_{xy} , S_x , S_y : moments, shear forces/unit width (global)
	Strain: ψ_x , ψ_y , ψ_{xy} , γ_{xz} , γ_{yz} : flexural, shear strains (global).
Modeller	See Results Tables (Appendix K) .

Local Axes

Not applicable (global axes are the reference).

Sign Convention

- ☐ [Standard plate element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Stiffness	Default.	2x2
	Fine.	As default.
Mass	Default.	2x2
	Fine.	As default.

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 55 Output strains as well as stresses.
- 105 Lumped mass matrix.
- 170 Suppress transfer of shape function arrays to disk.

Notes on Use

1. The element formulation involves imposing an assumed bi-linear shear strain field on the isoflex thin plate element QF4.
2. Though this element cannot model nonlinear behaviour, it can be mixed with other elements in a nonlinear analysis.
3. The element passes the [patch test](#) for convergence with rectangular and parallelogram element geometry.
4. The QF4,QF8,TF3,TF8 elements are usually more effective elements for thin plate analyses.

5. The QTF8 and TTF6 elements are usually more effective for thick plate analyses, and in such cases should be preferred to QSC4.
6. 3D solid elements should be used if the normal stress in the transverse direction is not insignificant in comparison with the in-plane stresses.
7. The following element combinations should be used for ribbed plates

Ribs with small or no eccentricity

- QSI4/TS3 elements with BMS3 elements,
- QTS4/TTS3 elements with BMS3 elements.

Ribs with large eccentricity

- QSL8/TSL6 elements with BSL3/BSL4/BXL4 elements,
- QTS4/TTS3 elements with BTS3 elements.

8. The through-thickness integration is performed explicitly.

Restrictions

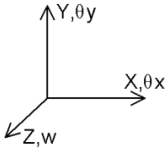
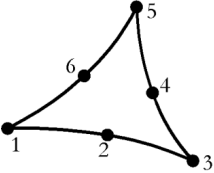
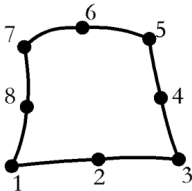
- ❑ [Avoid excessive aspect ratio](#)

Recommendations on Use

This element may be used to analyse any 2D plate type structures where transverse shear effects influence the solution, e.g. perforated thick plates.

2D Mindlin Thick Plate Flexure Element

General

Element Name	TTF6	QTF8
		
Element Group	Plates	
Element Subgroup	Mindlin Plates	
Element Description	A family of thick plate flexure elements based on a Mindlin plate formulation. The elements can accommodate curved boundaries and varying thicknesses. Transverse shear deformations are included.	
Number Of Nodes	6 or 8, numbered anticlockwise.	
Freedom	w, θ_x, θ_y : at each node.	
Node Coordinates	X, Y : at each node.	

Geometric Properties

$t_1 \dots t_n$ Thickness at each node.

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC THICK (Elastic: Orthotropic Thick)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC 5 (Elastic: Anisotropic Thick Plate)
	Rigidities.	RIGIDITIES 5 (Rigidities: Thick Plate)
Matrix	Not applicable	
Joint	Not applicable	
Concrete	Not applicable	

Elasto-Plastic	Not applicable
Creep	Not applicable
Damage	Not applicable
Viscoelastic	Not applicable
Shrinkage	Not applicable
Rubber	Not applicable
Generic Polymer	Not applicable
Composite	Not applicable

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. W , θ_x , θ_y : at nodes.
Concentrated Loads	CL	Concentrated loads. P_z , M_x , M_y : at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Uniformly distributed loads. W_z : normal pressure for element (global).
	FLD	Not applicable.
Body Forces	CBF	Constant body forces for element. Z_{cbf}
	BFP, BFPE	Body force potentials at nodes/for element. ϕ_1 , Z_{cbf}
Velocities	VELO	Velocities. V_z : at nodes.
Accelerations	ACCE	Accelerations. A_z : at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. M_x , M_y , M_{xy} , S_x , S_y : moments, shear forces/unit width (global). ψ_x , ψ_y , ψ_{xy} , γ_{xz} , γ_{yz} : flexural, shear strains /unit width (global).
	SSIG	Not applicable.
Residual Stresses	Not applicable.	
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. 0 , 0 , 0 , dT/dz , 0 , 0 , 0 , dTo/dz
Field Loads	Not applicable.	
Temp Dependent	Not applicable.	

Loads

Output

Solver Stress resultant: M_x , M_y , M_{xy} , S_x , S_y : moments, shear forces/unit width (global).
Strain: ψ_x , ψ_y , ψ_{xy} , γ_{xz} , γ_{yz} : flexural, shear strains /unit width (global).

Modeller See [Results Tables \(Appendix K\)](#).

Local Axes

Not applicable (global axes are the reference).

Sign Convention

☐ [Standard plate element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Stiffness	Default.	3-point (TTF6), 2x2 (QTF8)
	Fine (see <i>Options</i>).	3x3 (QTF8).
Mass	Default.	3-point (TTF6), 2x2 (QTF8)
	Fine (see <i>Options</i>).	3x3 (QTF8).

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes fine integration rule for element.
- 55** Output strains as well as stresses.
- 105** Lumped mass matrix.
- 170** Suppress transfer of shape function arrays to disk.

Notes on Use

1. The element formulations are based on an isoparametric approach. The variation of moments and shears within the element may be regarded as linear.
2. Though this element cannot model nonlinear behaviour, it can be mixed with other elements in a nonlinear analysis.
3. The elements pass the [patch test](#) for convergence with triangular and parallelogram element geometry.
4. These elements are usually more effective than the QSC4 thick shell element (section 7.6.2).
5. The elements tend to lock as the plate thickness approaches the thin plate limit since shear strain energy dominates the element stiffness. Therefore, a thin plate or shell element should be used when the depth/span ratio exceeds 1/50.
6. 3D solid elements should be used if the normal stress in the transverse direction is not insignificant in comparison with the in-plane stresses.
7. The following element combinations should be used for ribbed plates

Ribs with small or no eccentricity

- QSI4/TS3 elements with BMS3 elements,
- QTS4/TTS3 elements with BMS3 elements.

Ribs with large eccentricity

- QSL8/TSL6 elements with BSL3/BSL4/BXL4 elements,
 - QTS4/TTS3 elements with BTS3 elements.
8. The QTF8 element with 2*2 Gauss quadrature is generally more effective than the 3*3 rule. The 2*2 rule does, however, exhibit one zero energy mode which can be eliminated using option 18.
 9. The through-thickness integration is performed explicitly.

Restrictions

- ☐ [Ensure mid-side node centrality](#)
- ☐ [Avoid excessive element curvature](#)
- ☐ Avoid excessive aspect ratio

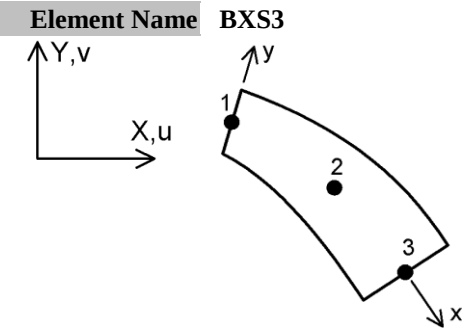
Recommendations on Use

These elements may be used to analyse any 2D plate type structures where transverse shear effects influence the solution, e.g. perforated thick plates.

Chapter 6 : Shell Elements.

2D Axisymmetric Thin Shell Element

General



Element Name	BXS3
Element Group	Shells
Element Subgroup	Axisymmetric Shells
Element Description	A parabolically curved axisymmetric thin shell element in 2D in which shear deformations are excluded. The geometric properties may vary along the length of the element.
Number Of Nodes	3.
End Releases	
End Freedoms	U, V, θ_z : at end nodes. dU: (relative local in-plane displacement) at the mid-length node.
Node Coordinates	X, Y: at each node.

Geometric Properties

t_1, t_2, t_3 Thickness at each node.

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC (Elastic: Orthotropic Plane Stress) MATERIAL PROPERTIES ORTHOTROPIC SOLID (Elastic: Orthotropic Thick)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC 2 (Not supported in LUSAS Modeller)
	Rigidities:	Not applicable.

Matrix	Not applicable.	
Joint	Not applicable.	
Concrete	Not applicable.	
Elasto-Plastic	Stress resultant:	MATERIAL PROPERTIES NONLINEAR 29 (Elastic: Isotropic, Plastic: Resultant) (ifcode not required)
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	Not applicable.
	Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
Creep		CREEP PROPERTIES (Creep)
	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic	Not applicable.	
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable.	
Generic	Not applicable	

Polymer
Composite Not applicable.

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, θ_z : at end nodes. dU: at the mid-length node.
Concentrated Loads	CL	Concentrated loads. Px, Py, Mx: point loads, moments/unit length/radian at end nodes (global). DPx: point load/unit length/radian at mid-length node (local).
Element Loads	ELDS	Element loads LTYPE, S1, Px, Py, Mx LTYPE=11: point loads and moments in local directions. LTYPE=12: point loads and moments in global directions. LTYPE, 0, Wx, Wy, Mx LTYPE=21: uniformly distributed loads in local directions. LTYPE=22: uniformly distributed loads in global directions. LTYPE=23: uniformly distributed projected loads in global directions LTYPE, S1, Wx1, Wy1, Mx1, S2, Wx2, Wy2, Mx2 LTYPE=31: distributed loads in local directions. LTYPE=32: distributed loads in global directions. LTYPE=33: distributed projected loads in global directions LTYPE, S1, Wx, Wy, Mx LTYPE=41: trapezoidal loads in local directions. LTYPE=42: trapezoidal loads in global directions. LTYPE=43: trapezoidal projected loads in global directions
Distributed Loads	UDL	Uniformly distributed loads. Wx, Wy: forces/unit length/radian in local x, y directions for element.
	FLD	Face Loads . Px, Py: local face pressures at nodes.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Ω_x , Ω_y , Ω_z , α_z
	BFP, BFPE	Body force potentials at nodes/for element. ϕ_1 , ϕ_2 , 0, 0, Xcbf, Ycbf

Velocities	VELO	Velocities. V_x, V_y : at nodes.
Accelerations	ACCE	Accelerations. A_x, A_y : at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. Resultants (for linear material models without cross section integration and material model 29). $N_x, N_\theta, M_x, M_\theta, 0$: axial and circumferential forces, moments/unit width. $\epsilon_x, \epsilon_\theta, \psi_x, \psi_\theta, 0$, axial and circumferential strains (all models).
	SSIG	Initial stresses/strains at Gauss points. (1) Resultants (for linear material models without cross section integration and material model 29). $N_x, N_\theta, M_x, M_\theta, 0$: axial and circumferential forces, moments/unit width. $\epsilon_x, \epsilon_\theta, \psi_x, \psi_\theta, 0$: axial and circumferential strains (all models). (2) Components (for linear material models with cross section integration and all nonlinear material models except 29). $0, 0, 0, 0, 0, 0, 0, 0, 0, 0, (\sigma_x, \sigma_\theta, \epsilon_x, \epsilon_\theta)$ Bracketed terms repeated for each fibre integration point.
Residual Stresses	SSR, SSRE	Not applicable.
	SSRG	Residual stresses at Gauss points. (1) Resultants (model 29). $N_x, N_\theta, M_x, M_\theta, 0$ (2) Components (all models except 29) $0, 0, 0, 0, 0, 0, 0, 0, 0, 0, (\sigma_x, \sigma_\theta)$ Bracketed terms repeated for each fibre integration point.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. $T, 0, dT/dy, 0, T_0, 0, dT_0/dy, 0$: in local directions.
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Force. $N_x, N_\theta, M_x, M_\theta$: axial and circumferential forces, moments/unit width in local directions. Strain. $\epsilon_x, \epsilon_\theta, \gamma_x, \gamma_\theta$: axial and circumferential strains. Layer stress and strain output is also available when using the nonlinear

continuum material models.

Modeller See [Results Tables \(Appendix K\)](#).

Local Axes

The local x-axis lies along the line of the element in the direction in which the nodes are numbered. The local y and z-axes form a right-hand set with the local x-axis such that the y-axis lies in the global XY-plane with the z-axis parallel to the global Z-axis.

Sign Convention

- ☐ Standard shell element. Axial and circumferential moments are positive for tension on element top fibre (the top fibre lies on the positive local y side of the element).

Formulation

Geometric Nonlinearity

Total Lagrangian	For large displacements, rotations up to 1 radian, and small strains.
Updated Lagrangian	For large displacements, rotation increments up to 1 radian and small strains.
Eulerian	Not applicable.
Co-rotational	Not applicable.

Integration Schemes

Stiffness	Default.	2-point.
	Fine (see <i>Options</i>).	3-point.
Mass	Default.	2-point.
	Fine (see <i>Options</i>).	3-point.

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes fine integration rule for element
- 47** X-axis taken as axis of symmetry
- 54** Updated Lagrangian geometric nonlinearity.

- 55 Output strains as well as stresses.
- 87 Total Lagrangian geometric nonlinearity
- 105 Lumped mass matrix.
- 157 Material model 29 (non cross-section elements), see *Notes*.
- 170 Suppress transfer of shape function arrays to disk.

Notes on Use

1. The element formulation is based on a constrained super-parametric approach.
2. The variation of axial force and moment along the length of the element is linear. The variation of displacements is cubic in the local y-direction, and quadratic in the local x direction.
3. Temperature dependent properties cannot be used with material model 29.
4. The through-thickness integration is performed explicitly for linear and stress resultant plasticity models and with a 5-point [Newton-Cotes](#) rule for all other material models.

Restrictions

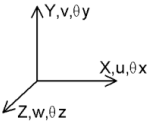
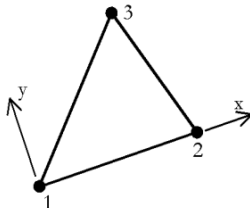
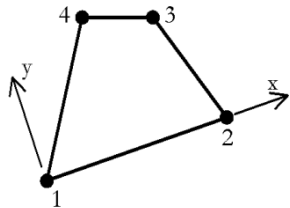
- ☐ [Ensure mid-side node centrality](#)
- ☐ Avoid excessive element curvature

Recommendations on Use

The element can be used for analysing shell structures which are axisymmetric, e.g. pressure vessels or pipes.

3D Flat Thin Shell Elements

General

Element Name	TS3	QSI4
		
Element Group	Shells	
Element Subgroup	Flat Thin Shells	
Element Description	A family of flat thin shells in 3D which include a high performance incompatible model. The elements take into account both membrane and flexural deformations. As required by thin plate theory, transverse shearing deformations are excluded. An average thickness value for each element is obtained from the specified nodal thicknesses. Since the elements are formulated in local element axes, directional material properties may be defined relative to the element orientation.	
Number Of Nodes	3 or 4 numbered anticlockwise.	
Freeloms	U, V, W, θ_x , θ_y , θ_z : at each node.	
Node Coordinates	X, Y, Z: at each node.	

Geometric Properties

Ez, t1... tn [Eccentricity](#) and thickness at each node.

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC (Elastic: Orthotropic Plane Stress) MATERIAL PROPERTIES ORTHOTROPIC SOLID (Elastic: Orthotropic Thick)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC 3 (Elastic: Anisotropic Thin Plate)
	Rigidities.	RIGIDITIES 6 (Rigidities: Shell) (D7, D8, D9, D11,

D12, D13, D16, D17, D18=0)

Matrix	Not applicable
Joint	Not applicable
Concrete	Not applicable
Elasto-Plastic	Not applicable
Creep	Not applicable
Damage	Not applicable
Viscoelastic	Not applicable
Shrinkage	
Rubber	Not applicable
Generic Polymer	Not applicable
Composite	Not applicable

SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W, θ_x , θ_y , θ_z : at nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py, Pz, Mx, My, Mz: at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Uniformly distributed loads. Wx, Wy, Wz: local surface pressures for element.
	FLD	Not applicable.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf
	BFP, BFPE	Body force potentials at nodes/for element. ϕ_1 , ϕ_2 , ϕ_3
Velocities	VELO	Velocities. Vx, Vy, Vz: at nodes.
Accelerations	ACCE	Accelerations. Ax, Ay, Az: at nodes.
Initial	SSI, SSIE	Initial stresses/strains at nodes/for element.

Stress/Strains		Resultants. N_x , N_y , N_{xy} , M_x , M_y , M_{xy} : forces, moments/unit width in local directions. ϵ_x , ϵ_y , γ_{xy} , ψ_x , ψ_y , ψ_{xy} : membrane, flexural strains in local directions.
	SSIG	Not applicable.
Residual Stresses		
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T , 0, 0, dT/dz , T_0 , 0, 0, dT_0/dz : in local directions.
Field Loads		Not applicable.
Temp Dependent Loads		Not applicable.

LUSAS Output

Solver	Stress resultant: N_x , N_y , N_{xy} , M_x , M_y , M_{xy} : forces, moments/unit width in local directions. Stress (default): σ_x , σ_y , σ_{xy} , σ_{max} , σ_{min} , β , σ_e : in local directions (see <i>Notes</i>). Strain: ϵ_x , ϵ_y , γ_{xy} , ψ_x , ψ_y , ψ_{xy} : membrane, flexural strains in local directions.

Modeller See [Results Tables \(Appendix K\)](#).

Local Axes

- ☐ [Standard area element](#)

Sign Convention

- ☐ [Thin shell element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Stiffness	Default.	1-point for the in-plane incompatible modes, (QSI4),

		2x2 for the in-plane compatible modes, (QSI4), 2x2 for bending (QSI4). 1-point for in-plane (TS3), 3-point for bending (TS3).
	Fine.	As default.
Mass	Default.	1-point for the in-plane incompatible modes, (QSI4), 2x2 for the in-plane compatible modes, (QSI4), 2x2 for bending (QSI4). 1-point for in-plane (TS3), 3-point for bending (TS3).
	Fine.	As default.

Mass Modelling

Lumped mass only.

Options

- 32 Suppress stress output but not stress resultants.
- 34 Outputs stress resultants.
- 55 Outputs strains as well as stresses.
- 59 Outputs local direction cosines for elements.
- 170 Suppresses transfer of shape function arrays to disk.

Notes on Use

1. The element formulations are based on the standard [isoflex](#) approach for the flexural matrices.
2. The variation of membrane stresses within the element can be regarded as constant for TS3 and linear for QSI4. The higher order membrane performance of QSI4 is due to the addition of four incompatible in-plane displacement modes. The variation of flexural stresses can be regarded as linear for all elements.
3. The stress results are most easily interpreted if the local element axes are all parallel.
4. The elements pass the [patch test](#) for mixed triangular and quadrilateral geometry.
5. Stress output to the LUSAS output file is on 4 lines:
 - Stresses due to membrane action.
 - Top surface stresses due to bending action.

- Top surface stresses due to membrane and bending action.
- Bottom surface stresses due to membrane and bending action.

Gauss point output is not available.

6. All distributed loading will be lumped at the nodes.
7. For effective analysis of curved shell structures, a flat shell element should not extend over more than 15 degrees of arc.
8. Though this element cannot model nonlinear behaviour, it can be mixed with other elements in a nonlinear analysis.
9. A system variable is used to alter the artificial stiffness for in-plane rotations.
10. A fine discretisation will be required to reproduce the correct behavioural response for curved structures. Therefore, the Semiloof shell elements (QSL8,TSL6) or the thick shell elements (QTS8, TTS6) may be more appropriate.
11. The ORTHOTROIC SOLID material model may be used with either composite or non-composite thin shell elements. Using a Solid rather than a Thick orthotropic material means that a local coordinate may be used to orientate the material.

Restrictions

- ☐ [Avoid excessive aspect ratio.](#)
- ☐ [Avoid excessive warping.](#)

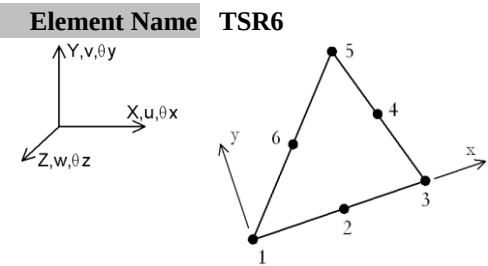
Recommendations on Use

- The flat thin shell elements are suitable for modelling both flat and curved thin shell structures which exhibit negligible transverse shear deformations.
- A fine discretisation will be required to reproduce the correct behavioural response for curved structures. Therefore, the Semiloof shell elements (QSL8,TSL6) or the thick shell elements (QTS8, TTS6) may be more appropriate.
- The Semiloof shell elements (QSL8,TSL6) or the thick shell elements (QTS8, TTS6) are more effective for structures containing multiple shell intersections.
- The Semiloof shell elements (QSL8,TSL6) or the thick shell elements (QTS4, QTS8, TTS3, TTS6) may be more effective for eigen-analyses since a consistent mass matrix is available.
- The Semiloof shell elements (QSL8,TSL6) should be utilised for nonlinear analyses.

- The elements can be combined with BMS3 beam elements for analysing ribbed shells with small or no eccentricity. However, the Semiloof shell (QSL8,TSL6) and beam (BSL3,BSL4,BXL4) are more effective for thin ribbed shells with larger eccentricity. For thick ribbed shells with larger eccentricity the thick shell (QTS4, QTS8, TTS3, TTS6) and co-rotational beam (BTS3) are recommended.

3D Flat Thin Nonlinear Shell Element

General



Element Group	Shells
Element Subgroup	Flat Thin Shells
Element Description	A triangular shell element for the analysis of faceted shell geometries, including multiple branched junctions. The elements can accommodate varying thickness and anisotropic material properties. The element is based on the “Morley shell” formulation and assumes constant membrane and bending strains across the element. As required by thin shell theory, transverse shearing deformations are excluded.
Number Of Nodes	6 numbered anticlockwise.
Freedom	U, V, W: at corner nodes. θ_1 : (loof rotation) at mid-side nodes (see <i>Notes</i>).
Node Coordinates	X, Y, Z: at each node.

Geometric Properties

$t_1 \dots t_n$ Thickness at each node.

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC (Elastic: Orthotropic Plane Stress) MATERIAL PROPERTIES ORTHOTROPIC SOLID (Elastic: Orthotropic Thick)

	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC 3 (Elastic: Anisotropic Thin Plate)
	Rigidities:	RIGIDITIES 6 (Rigidities: Shell)
Matrix	Not applicable	
Joint	Not applicable	
Concrete		MATERIAL PROPERTIES NONLINEAR 94 (Elastic: Isotropic, Plastic: Multi-Crack Concrete)
		MATERIAL PROPERTIES NONLINEAR 102 (Elastic: Isotropic, Plastic: Smoothed Multi-Crack Concrete)
Elasto-Plastic	Stress resultant:	MATERIAL PROPERTIES NONLINEAR 29 (Elastic: Isotropic, Plastic: Resultant) (ifcode not required)
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Volumetric Crushing:	Not applicable.
	Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
Creep		CREEP PROPERTIES (Creep)

	CEB-FIP	Not applicable.
	Chinese	Not applicable.
	Eurocode	Not applicable.
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic	Not applicable	
Shrinkage		GENERAL, USER
Rubber	Not applicable.	
Generic Polymer	Not applicable	
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W: at corner nodes. θ_1 : at mid-side nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py, Pz: at corner nodes. M1: at mid-side nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Uniformly distributed loads. Wx, Wy, Wz: mid-surface local pressures for element.
Body Forces	FLD	Not applicable.
	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf, $\Omega_x, \Omega_y, \Omega_z, \alpha_x, \alpha_y, \alpha_z$
	BFP, BFPE	Body force potentials at nodes/for element. $\phi_1, \phi_2, \phi_3, 0$, Xcbf, Ycbf, Zcbf, where ϕ_1, ϕ_2, ϕ_3 are the face loads in the local coordinate system.

LUSAS Output

320

Local Axes

- [Standard area element](#)

Sign Convention

- ☐ [Thin shell element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	Not applicable.
Updated Lagrangian	Not applicable.
Eulerian	Not applicable.
Co-rotational	For large displacements and rotations

Integration Schemes

Stiffness Default.	1-point
Fine.	1-point
Coarse.	1-point
Mass Default.	1-point
Fine.	1-point

Mass Modelling

- ☐ Consistent mass.

Options

- 32 Suppresses stress output but not resultants.
- 34 Outputs element stress resultants.
- 55 Outputs strains as well as stresses.
- 59 Outputs local direction cosines at nodes and Gauss points.
- 77 Output principal stresses and directions.
- 139 Output yielded Gauss points only.

Notes on Use

1. The element formulations are based on a [Kirchhoff](#) hypothesis for thin shells.
2. The stresses are constant within the elements.

3. The loof rotations refer to rotations about the element edge at the mid-side nodes. The positive direction of a loof rotation is defined by a right-hand screw rule applied to a vector running in the direction of the lower to higher numbered corner nodes. It should be noted that this direction is enforced on a global level which means that the loof rotations along the adjoining edge of several elements will be consistent in terms of direction and ordering.
4. The element edges must remain straight even though the elements have mid-side nodes.
5. The elements pass the [patch test](#) for convergence.
6. Stresse will not be output when using RIGIDITIES or material model 29.
7. The through-thickness integration is performed explicitly for linear analyses and a 5-point [Newton-Cotes](#) rule is utilised for materially nonlinear analyses with continuum material models. The through-thickness integration rules are as follows:
 - Linear models: 3-layers.
 - Nonlinear models: 5-layers.

Restrictions

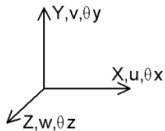
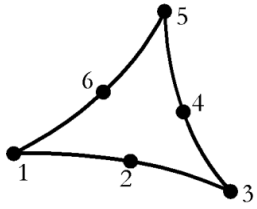
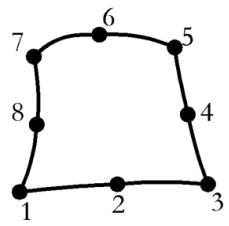
- ☐ [Ensure mid-side node centrality and straight element edges](#)
- ☐ Avoid excessive aspect ratio

Recommendations on Use

- These elements may be utilised for analysing flat and faceted 3D shell structures where the transverse shear effects do not influence the solution. The configuration of the nodal freedoms provides an element suitable for modelling intersecting shells.
- The elements are recommended for geometrically nonlinear problems where large displacements and rotations occur. The single Gauss point integration scheme gives rise to a computationally efficient solution, however, the mesh may need to be refined if there is an unacceptable differentiation in stresses between adjacent elements..

Semiloof Curved Thin Shell Elements

General

Element Name	TSL6	QSL8
		
Element Group	Shells	
Element Subgroup	Semiloof Shells	
Element Description	A family of shell elements for the analysis of arbitrarily curved shell geometries, including multiple branched junctions. The elements can accommodate generally curved geometry with varying thickness and anisotropic and composite material properties. The element formulation takes account of both membrane and flexural deformations. As required by thin shell theory, transverse shearing deformations are excluded.	
Number Of Nodes	6 or 8 numbered anticlockwise.	
Freedoms	U, V, W: at corner nodes. U, V, W, θ_1 , θ_2 : (loof rotations) at mid-side nodes (see <i>Notes</i>).	
Node Coordinates	X, Y, Z: at each node.	

Geometric Properties

$t_1 \dots t_n$ Thickness at each node. Also see *Composite Geometry* data chapter.

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC (Elastic: Orthotropic Plane Stress) MATERIAL PROPERTIES ORTHOTROPIC SOLID (Elastic: Orthotropic Solid)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC 3 (Elastic: Anisotropic Thin Plate)

	Rigidities.	RIGIDITIES 6 (Rigidities: Shell)
Matrix	Not applicable	
Joint	Not applicable	
Concrete		MATERIAL PROPERTIES NONLINEAR 94 (Elastic: Isotropic, Plastic: Multi-Crack Concrete)
		MATERIAL PROPERTIES NONLINEAR 102 (Elastic: Isotropic, Plastic: Smoothed Multi-Crack Concrete)
Elasto-Plastic	Stress resultant:	MATERIAL PROPERTIES NONLINEAR 29 (Elastic: Isotropic, Plastic: Resultant) (ifcode not required)
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Volumetric Crushing:	Not applicable.
Creep	Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises Orthotropic: Hill, Hoffman)
		CREEP PROPERTIES (Creep)
	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)

	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic	Not applicable	
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable.	
Generic Polymer	Not applicable	
Composite	Composite shell:	COMPOSITE PROPERTIES
Loading		
Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W: at corner nodes. U, V, W, θ_1 , θ_2 : at mid-side nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py, Pz: at corner nodes. Px, Py, Pz, M1, M2: at mid-side nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL FLD	Uniformly distributed loads. Wx, Wy, Wz: mid-surface local pressures for element. Not applicable.
Body Forces	CBF BFP, BFPE	Constant body forces for element. Xcbf, Ycbf, Zcbf, Ω_x , Ω_y , Ω_z , α_x , α_y , α_z Body force potentials at nodes/for element. ϕ_1 , ϕ_2 , ϕ_3 , 0, Xcbf, Ycbf, Zcbf, where ϕ_1 , ϕ_2 , ϕ_3 are the face loads in the local coordinate system.
Velocities	VELO	Velocities. Vx, Vy, Vz: at nodes.
Accelerations	ACCE	Accelerations. Ax, Ay, Az: at nodes.

Initial Stress/Strains	SSI, SSIE	Not applicable.
	SSIG	Initial stresses/strains at Gauss points. (1) Resultants (for linear analysis and model 29) N_x , N_y , N_{xy} , M_x , M_y , M_{xy} , ϵ_x , ϵ_y , γ_{xy} , ψ_x , ψ_y , ψ_{xy} : forces, moments/unit width and membrane/flexural strains in local directions. (2) Components (for all other nonlinear material models) are: 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, $(\sigma_x, \sigma_y, \sigma_{xy}, \epsilon_x, \epsilon_y, \gamma_{xy})$ - with the bracketed terms repeated for each of the five layers. (See note 7 in the Notes of Use) section.
Residual Stresses	SSR, SSRE	Not applicable.
	SSRG	Residual stresses at Gauss points. (1) Resultants (for model 29) N_x , N_y , N_{xy} , M_x , M_y , M_{xy} : forces, moments/unit width in local directions. (2) Components (for all other nonlinear material models) are: 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, $(\sigma_x, \sigma_y, \sigma_{xy})$ - with the bracketed terms repeated for each of the five layers. (See note 7 in the Notes of Use) section.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, dT/dz, To, 0, 0, dTo/dz
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

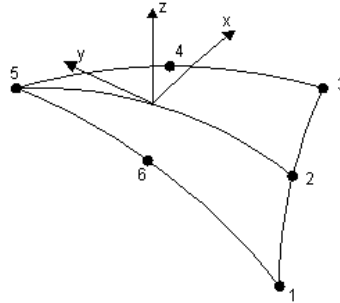
LUSAS Output

Solver	Stress resultant: N_x , N_y , N_{xy} , M_x , M_y , M_{xy} : forces, moments/unit width in local directions. Stress (default): σ_x , σ_y , σ_{xy} , σ_{max} , σ_{min} , β , σ_e : in local directions (see <i>Notes</i>). Strain: ϵ_x , ϵ_y , γ_{xy} , ψ_x , ψ_y , ψ_{xy} : membrane, flexural strains in local directions.
Modeller	See Results Tables (Appendix K) .

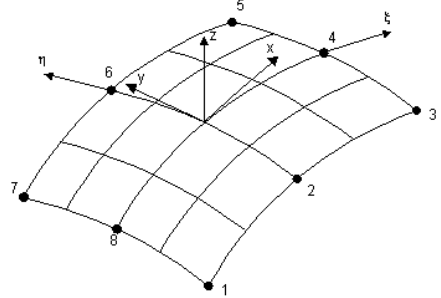
Local Axes

- **Local y axis** The local element y-axis at a point coincides with a curvilinear line $\xi = \text{constant}$ in the natural coordinate system which lies in the shell mid-surface.
- **Local x axis** The local x-axis at a point is perpendicular to the local y-axis in the positive η direction and is tangential to the shell mid-surface.
- **Local z axis** The local z-axis forms a right-hand set with the x and y axes and the direction is given by the ordering of the element nodes according to a right-hand screw rule. The local z-axis +ve direction defines the element top surface.

TSL6



QSL8



Sign Convention

- Thin shell element (see Notes).

Formulation

Geometric Nonlinearity

Total Lagrangian	For large displacements, rotations up to 1 radian and small strains.
Updated Lagrangian	For large displacements, rotation increments up to 1 radian and small strains.
Eulerian	Not applicable.
Co-rotational	Not applicable.

Integration Schemes

Stiffness	Default.	3-point (TSL6), 5-point (QSL8).
	Fine (see	3x3 (QSL8)

	<i>Options).</i>	
	Coarse (see	2x2 (QSL8)
	<i>Options).</i>	
Mass	Default.	3-point (TSL6), 5-point (QSL8).
	Fine (see	3x3 (QSL8)
	<i>Options).</i>	

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes fine integration rule.
- 19** Invokes coarse integration rule.
- 32** Suppresses stress output but not resultants.
- 34** Outputs element stress resultants.
- 54** Updated Lagrangian geometric nonlinearity.
- 55** Outputs strains as well as stresses
- 59** Outputs local direction cosines at nodes and Gauss points.
- 87** Total Lagrangian geometric nonlinearity.
- 102** Switch off load correction stiffness due to centripetal acceleration.
- 105** Lumped mass matrix.
- 138** Output yield flags only.
- 139** Output yielded Gauss points only.
- 169** Suppress extrapolation of stresses to nodes.
- 170** Suppress transfer of shape function arrays to disk.

Notes on Use

1. The element formulations are based on a [Kirchhoff](#) hypothesis for thin shells.
2. The variation of stresses within the elements may be regarded as linear.
3. The loof rotations refer to rotations about the element edge at the loof points. The positive direction of a loof rotation is defined by a right-hand screw rule applied to a vector running in the direction of the lower to higher numbered corner nodes. It should be noted that this direction is enforced on a global level which means that the loof rotations along the adjoining edge of several elements will be consistent in terms of direction and ordering. The ordering is such that loof point 1 is located between the lower numbered node and the appropriate mid-side node. Similarly loof point 2 lies between the mid-side

- node and the higher numbered node along an element edge. The loof rotations are actually specified at the element mid-side nodes.
4. The elements pass the [patch test](#) for convergence for mixed triangular and quadrilateral element geometry.
 5. Stress output to the LUSAS output file is on 4 lines:
 - Stresses due to membrane action.
 - Top surface stresses due to bending action.
 - Top surface stresses due to membrane and bending action.
 - Bottom surface stresses due to membrane and bending action.
 6. Stresses will not be output when using RIGIDITIES or material model 29. Averaged stresses will not be processed when using RIGIDITIES.
 7. The through-thickness integration is performed explicitly for linear analyses and a 5-point [Newton-Cotes](#) rule is utilised for materially nonlinear analyses with continuum material models. The through-thickness integration rules are as follows:
 - Linear models: 3-layers.
 - Nonlinear models: 5-layers.
 - Composite model: Variable.
 8. The quadrature points of the 3-point rule are non-standard.
 9. The coarse 2*2 quadrature rule provides the most effective element if the mesh is highly constrained. However, the element possesses two mechanisms, the usual in-plane hourglass mechanism encountered when reduced integration is utilised with 8-noded elements and an out of plane mechanism. The in-plane mechanism is rarely activated but the out-of-plane mechanism may be more troublesome, particularly where elements are regular and have one zero principal curvature, e.g. a cylinder subject to internal pressure. Provided the mechanisms are not activated the element with 2*2 provides the best results.
 10. The 5-point quadrature rule provides an element with a performance below that of the element with 2*2 quadrature, but considerably better than the element with 3*3 quadrature. However, the element possesses a 'near' mechanism which may be activated for lightly constrained meshes, particularly if out of plane loads are present.
 11. The middle integration point of the 5 point rule is only implemented as a method of reducing the excitation of spurious modes (or mechanisms) which are present with the 2*2 integration rule. The 5th integration point is actually

weighted with an arbitrarily small value which has the effect of stabilising the results. For these reasons, values from the middle integration point are not taken into account for the nodal extrapolation.

12. The 3*3 quadrature rule provides an element that has no mechanisms but tends to provide over-stiff solutions. Therefore, a finer discretisation is required than if the 5-point quadrature rule is used.

Restrictions

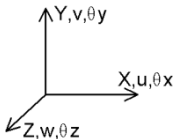
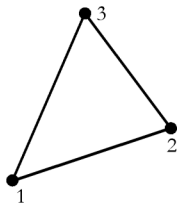
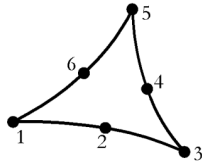
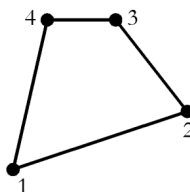
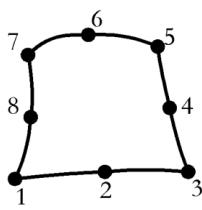
- ☐ [Ensure mid-side node centrality](#)
- ☐ [Avoid excessive element curvature](#)
- ☐ Avoid excessive aspect ratio

Recommendations on Use

- These elements may be utilised for analysing flat and curved 3D shell structures where the transverse shear effects do not influence the solution. The configuration of the nodal freedoms provides an element suitable for modelling intersecting shells, e.g. tubular joints and also for use with solid elements (HX20).
- The elements may be combined with the Semiloof beam (BSL3,BSL4,BXL4) for analysing ribbed plates and shells.

3D Thick Shell Elements

General

Element Name	TTS3	TTS6
		
	QTS4	QTS8
		
Element Group	Shells	
Element Subgroup	Thick Shells	
Element Description	A family of shell elements for the analysis of arbitrarily thick and thin curved shell geometries, including multiple branched junctions. The quadratic elements can accommodate generally curved geometry while all elements account for varying thickness. Anisotropic and composite material properties can be defined. These degenerate continuum elements are also capable of modelling warped configurations. The element formulation takes account of membrane, shear and flexural deformations. The quadrilateral elements use an assumed strain field to define transverse shear which ensures that the element does not lock when it is thin (see <i>Notes</i>).	
Number Of Nodes	3, 4, 6 or 8 numbered anticlockwise.	
Freedom	Default: 5 degrees of freedom are associated with each node U, V, W, $\theta\alpha$, $\theta\beta$. To avoid singularities, the rotations $\theta\alpha$ and $\theta\beta$ relate to axes defined by the orientation of the normal at a node, see Thick Shell Nodal Rotation. These rotations may be transformed to relate to the global axes in some instances (see <i>Notes</i>). Degrees of freedom relating to global axes: U, V, W, θ_x, θ_y, θ_z may be enforced using the Nodal Freedom data input, or for all shell nodes	

		by using option 278 (see <i>Notes</i>).
	Node	X, Y, Z: at each node.
	Coordinates	
	Nodal	5 or 6.
	Freedom	

Geometric Properties

e_z , t_1 ... t_n [Eccentricity](#) and thickness at each node.

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC THICK (Elastic: Orthotropic Thick)
		MATERIAL PROPERTIES ORTHOTROPIC SOLID (Elastic: Orthotropic Thick)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC 5 (Elastic: Anisotropic Thick Plate)
	Rigidities.	Not applicable.
Matrix	Not applicable	
Joint	Not applicable	
Concrete		MATERIAL PROPERTIES NONLINEAR 94 (Elastic: Isotropic, Plastic: Multi-Crack Concrete)
		MATERIAL PROPERTIES NONLINEAR 102 (Elastic: Isotropic, Plastic: Smoothed Multi-Crack Concrete)
Elasto-Plastic	Stress resultant:	Not applicable.
	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)
	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Volumetric Crushing:	Not applicable.
	Stress Potential	STRESS POTENTIAL VON_MISES, HILL, HOFFMAN (Isotropic: von Mises, Modified von Mises)

		Orthotropic: Hill, Hoffman)
Creep		CREEP PROPERTIES (Creep)
	CEB-FIP	MATERIAL PROPERTIES NONLINEAR 86 CEB-FIP (Concrete creep model to CEB-FIP Model Code 1990)
	Chinese	MATERIAL PROPERTIES NONLINEAR 86 CHINESE (Concrete creep model to Chinese Code of Practice)
	Eurocode	MATERIAL PROPERTIES NONLINEAR 86 EUROCODE (Concrete creep model to EUROCODE_2)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic	Not applicable	
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable	
Generic	Not applicable	
Polymer		
Composite	Composite shell:	COMPOSITE PROPERTIES

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. 5 degrees of freedom: U, V, W, $\theta\alpha$, $\theta\beta$ or 6 degrees of freedom: U, V, W, θ_x , θ_y , θ_z
Concentrated Loads	CL	Concentrated loads. 5 degrees of freedom: Px, Py, Pz, $M\alpha$, $M\beta$, where $M\alpha$ and $M\beta$ relate to axes defined by $\theta\alpha$ and $\theta\beta$ respectively. 6 degrees of freedom: Px, Py, Pz, Mx, My, Mz.
Element Loads	Not applicable.	
Distributed Loads	UDL	Uniformly distributed loads. Wx, Wy, Wz: mid-surface local pressures for element.
	FLD	Not applicable.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf, Ω_x , Ω_y , Ω_z , α_x , α_y , α_z
	BFP, BFPE	Body force potentials at nodes/for element. ϕ_1 , ϕ_2 , ϕ_3 , 0, Xcbf, Ycbf, Zcbf, where ϕ_1 , ϕ_2 , ϕ_3 are the face loads in the local coordinate system.

Velocities	VELO	Velocities. V_x , V_y , V_z : at nodes.
Accelerations	ACCE	Accelerations. A_x , A_y , A_z : at nodes.
Initial Stress/Strains	SSI, SSIE	Not applicable.
	SSIG	Initial stresses/strains at Gauss points. Stress/strain components relating to local axes at Gauss points: σ_x , σ_y , σ_{xy} , σ_{yz} , σ_{xz} , ϵ_x , ϵ_y , γ_{xy} , γ_{yz} , γ_{xz} . All of these 10 terms are repeated for each fibre integration point through the thickness (see <i>Notes</i>).
Residual Stresses	SSR, SSRE	Not applicable.
	SSRG	Residual stresses at Gauss points. Stress components relating to local axes at Gauss points: σ_x , σ_y , σ_{xy} , σ_{yz} , σ_{xz} all of these 5 terms are repeated for each fibre integration point through the thickness (see <i>Notes</i>).
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T , 0, 0, dT/dz , T_0 , 0, 0, dT_0/dz
Field Loads	Not applicable.	
Temp Dependent Loads	Temp Not applicable.	

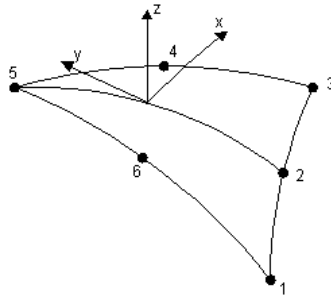
LUSAS Output

Solver	Stress resultant: N_x , N_y , N_{xy} , M_x , M_y , M_{xy} , S_x , S_y : forces, moments/unit width in local directions. Stress (default): σ_x , σ_y , σ_{xy} , σ_{yz} , σ_{xz} , σ_e : in local directions (see <i>Notes</i>). Strain: ϵ_x , ϵ_y , γ_{xy} , γ_{yz} , γ_{xz} , ϵ_e : in local directions (see <i>Notes</i>).
Modeller	See Results Tables (Appendix K) .

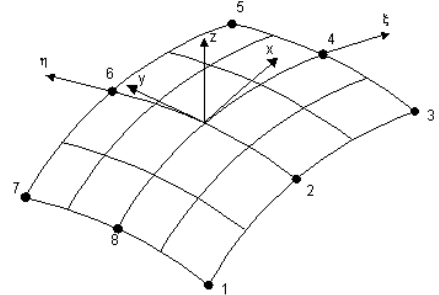
Local Axes

The local element x-axis at a point coincides with a curvilinear line $\eta = \text{constant}$ in the natural coordinate system which lies in the shell mid-surface. The local z-axis at a point is obtained from the cross product of a curvilinear line $\xi = \text{constant}$ in the natural coordinate system and the local x-axis. The local y-axis forms a right-hand set with the x and z axes and the direction is given by the ordering of the element nodes according to a right-hand screw rule. The local z-axis +ve direction defines the element top surface.

TTS6



QTS8



Sign Convention

- ☐ Thick shell element (see *Notes*).

Formulation

Geometric Nonlinearity

- Total Lagrangian** For large displacements, large rotations and small strains.
Updated Lagrangian Not applicable.
Eulerian Not applicable.
Co-rotational Not applicable.

Integration Schemes

- Stiffness** Default. 1-point (TTS3), 3-point (TTS6), 2x2 (QTS4, QTS8).
 Fine (see *Options*). 3-point (TTS3), 5-point (QTS8)
Mass Default. 1-point (TTS3), 3-point (TTS6), 2x2 (QTS4, QTS8).
 Fine (see *Options*). 3-point (TTS3), 5 point (QTS8)

Mass Modelling

- ☐ Consistent mass (default).
☐ Lumped mass.

Options

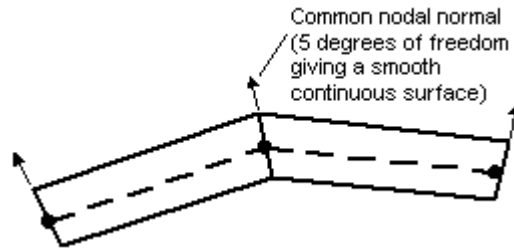
- 18** Invokes fine integration rule.
32 Suppresses stress output but not resultants.
34 Outputs element stress resultants.
55 Outputs strains as well as stresses

- 59 Outputs local direction cosines at nodes and Gauss points.
- 77 Outputs principal stresses.
- 87 Total Lagrangian geometric nonlinearity.
- 102 Switch off load correction stiffness due to centripetal acceleration.
- 105 Lumped mass matrix.
- 139 Output yielded Gauss points only.
- 169 Suppress extrapolation of stresses to nodes.
- 171 Switch off assumed strain field for QTS4 elements.
- 278 Six degrees of freedom
- 396 Invokes the improved transverse shear calculation ('on' by default for models created by version 14.4 and above, and 'off' - for models created by previous versions).

Notes on Use

1. For TTS3 elements all moments and shears are constant for the element. For QTS4 the variations of moments, out of plane shears and in-plane loads is near-constant and the variation of in-plane shear is near-linear. For TTS6 and QTS8 elements the variation of moments and in-plane shear is near-linear while the variation of out of plane shears is near constant.
2. The QTS8 element fails the shear [patch test](#) when the assumed strain field is utilised with 2*2 or 5 point integration rule. When carrying out analyses involving these elements that are dominated by transverse shear effects, e.g. a shear wall, it is recommended that the assumed strain field is disabled. This is the default setting for QTS8 elements. Option 110 may be used to invoke the assumed strain interpolation but this is not recommended for general use.
3. The assumed strain field is invoked automatically for QTS4 elements. The assumed strain field may be revoked for QTS4 by specifying Option 171.
4. Continuum stresses (and strains using Option 55) at each fibre integration point are output by default. For linear materials these stresses relate to the top, middle and bottom surfaces of the element. If a nonlinear material is specified then stresses are output at 5 points through the thickness after material yield.
5. Option 55 must be specified if nonlinear state variables are to be written to the LUSAS output file.
6. The through-thickness integration rules are as follows:
 - Linear material models: 3-layers.
 - Nonlinear material models: 5-layers.
 - Composite model: variable.

7. Initial stresses/strains must be specified at 3 layers for a linear material or 5 layers for a nonlinear material. Residual stresses must be specified for 5 layers. In all instances the stresses/strains are specified sequentially from the bottom surface to the top.
8. There are usually 2 rotational degrees of freedom and a common nodal normal associated with each node giving a smooth surface to the shell assembly:



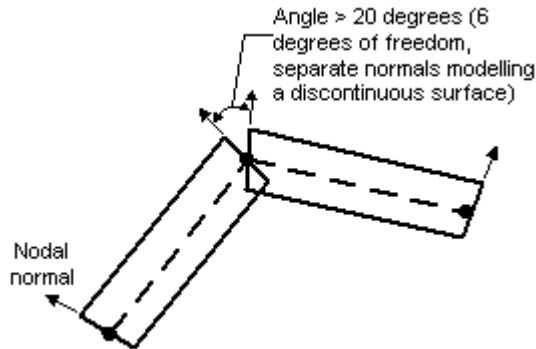
The direction of the axes defining the rotations depends upon the orientation of the normal at a node (see [Thick Shell Nodal Rotation](#)). In certain circumstances 3 rotational degrees of freedom relating to global axes will be assigned to a node. This is done automatically:

- When connecting with beam elements, joint elements or other types of shells, eg.QS14.
- When a Concentrated Load is applied in LUSAS Modeller.
- When a Support is applied in LUSAS Modeller.
- When option 278 is specified.

If Option 278 is specified then all nodes for these shell element types will be assigned six global degrees of freedom. To overcome the problems associated with in-plane drilling rotations an artificial stiffness is automatically included for the rotation about the shell normal. The use of Option 278 is not recommended for analyses that involve large displacements or rotations. LUSAS Modeller will automatically specify Option 278 unless geometric nonlinearity has been defined.

Option 278 should be switched **off** if QTS4 elements are to be used to model thick curved shells in which membrane action leads to a significant difference between the in-plane strains in the top and bottom surface of the shell. If Option 278 is not disabled under these circumstances the moments associated with this in-plane strain differential are not accurately accounted for. An alternative approach would be to switch to QTS8 elements as these elements produce more accurate moments under these conditions.

When the maximum angle between adjacent normals at a node is greater than 20 degrees, e.g., branched shell structures. (20 degrees is a default value which may be changed using the SYSTEM parameter SHLANG); if the nodal freedom command has **not** been specified for that node.



9. A system variable (STFINP) is used to alter the artificial stiffness for in-plane rotations. This system parameter can only be used in conjunction with Option 278.
10. The desired number of rotational degrees of freedom for a node may be enforced through the NODAL FREEDOMS data input. Care must be taken if 6 degrees of freedom are specified in this manner as a singularity may occur if appropriate in-plane rotations are not restrained. This facility is provided together with the TRANSFORMED FREEDOMS data chapter to allow more flexibility in the specification of boundary conditions. In these circumstances, the in-plane rotation about the normal of the shell must usually be restrained to avoid singularities. In general, wherever possible, 5 degrees of freedom should be used when the shell surface is smooth.
11. When a nonlinear material is used with this element the transverse shear stresses are excluded from the plasticity computations i.e. the transverse shear stresses are assumed to remain elastic. This means that if a nonlinear material is used in applications where transverse shear tends to dominate the stress field the equivalent von Mises and maximum principal stresses can exceed the uniaxial yield stress.
12. The TTS3 and QTS8 elements possess one out of plane mechanism when using the default integration rules. The 3 noded element is most effective using the one point rule.
13. The through-thickness integration is performed by utilising a 3 point [Newton-Cotes](#) rule for linear materials and a 5 point rule for nonlinear materials and creep. In an analysis involving material nonlinearity, a 3 point rule is used until the material yields and then a 5 point rule is invoked.

14. The thick shell formulation assumes constant transverse shear deformation. In the post-processing stage, after the application of the constitutive relationship, this results in a constant transverse shear stress. This result can be improved by taking into account the true parabolic shear stress distribution while preserving the same shear resultant. Thus, when Option 396 is used, the transverse shear stresses for a non-layered shell are set to zero at the top and bottom and to 1.5 times the constant value at the middle. For a layered shell, the distribution of the transverse shear depends on the in-plane stiffness of the layers. The output results are for the middle of the layer, thus the top and bottom layers will not have zero transverse shear.
15. The ORTHOTROPIC SOLID material model may be used with either composite or non-composite thick shell elements. Using a Solid rather than a Thick orthotropic material means that a local coordinate may be used to orientate the material.

Restrictions

- ☐ Ensure mid-side node centrality
- ☐ Avoid excessive element curvature
- ☐ Avoid excessive aspect ratio

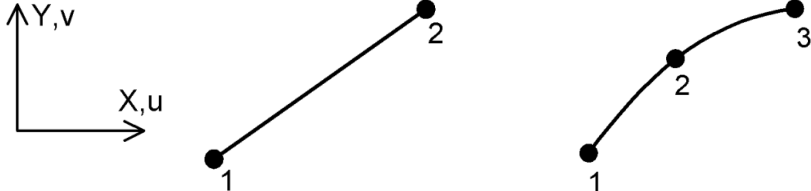
Recommendations on Use

- These elements may be utilised for analysing flat and curved 3D shell structures where it is necessary to account for transverse shear. This typically involves thick shell structures where transverse shear deformation can have a considerable influence on the response. The degenerate continuum formulation also allows the low order quadrilateral element (QTS4) to successfully model warped shell configurations.
- The elements may be used for modelling intersecting shells or branched shell junctions. In this instance the nodal rotation freedoms are transformed to relate to the global axes. For modelling stiffened shell structures, the shells may be connected to beam elements BMS3 or BTS3.
- This family of thick shell elements offers a consistent formulation of the tangent stiffness which makes them particularly effective in geometrically nonlinear applications.
- Be aware that when the shell is defined with eccentricity to a reference surface and this reference surface does not pass through the centroid of the cross section, membrane forces or displacements prescribed/calculated at the nodes will cause bending.

Chapter 7 : Membrane Elements.

2D Axisymmetric Membrane Elements

General

Element Name	BXM2	BXM3
		
Element Group	Membranes	
Element Subgroup	Axisymmetric Membranes	
Element Description	Straight and curved axisymmetric membrane elements which can accommodate varying thickness. The formulations apply over a unit radian segment of the structure. The loading and boundary conditions are axisymmetric. The elements are numerically integrated. The default axis of symmetry is the Y-axis.	
Number Of Nodes	2 or 3.	
Freedom	U, V: at each node.	
Node Coordinates	X, Y: at each node.	

Geometric Properties

$t_1 \dots t_n$ Thickness at each node.

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
Matrix	Not applicable	
Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic	Tresca:	MATERIAL PROPERTIES NONLINEAR 61 (Elastic: Isotropic, Plastic: Tresca, Hardening: Isotropic Hardening Gradient, Isotropic Plastic Strain or Isotropic Total Strain)
	Drucker-Prager:	MATERIAL PROPERTIES NONLINEAR 64 (Elastic: Isotropic, Plastic: Drucker-Prager, Hardening: Granular)

	Mohr-Coulomb:	MATERIAL PROPERTIES NONLINEAR 65 (Elastic: Isotropic, Plastic: Mohr-Coulomb, Hardening: Granular with Dilation)
	Optimised Implicit Von Mises:	MATERIAL PROPERTIES NONLINEAR 75 (Elastic: Isotropic, Plastic: Von Mises, Hardening: Isotropic & Kinematic)
	Volumetric Crushing:	Not applicable.
	Stress Potential	STRESS POTENTIAL VON_MISES (Isotropic: von Mises, Modified von Mises)
Creep		CREEP PROPERTIES (Creep)
Damage		DAMAGE PROPERTIES SIMO, OLIVER (Damage)
Viscoelastic	Not applicable	
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Ogden:	MATERIAL PROPERTIES RUBBER OGDEN (Rubber: Ogden) (See Restrictions)
	Mooney-Rivlin:	MATERIAL PROPERTIES RUBBER MOONEY_RIVLIN (Rubber: Mooney-Rivlin) (See Restrictions)
	Neo-Hookean:	MATERIAL PROPERTIES RUBBER NEO_HOOKEAN (Rubber: Neo-Hookean) (See Restrictions)
	Hencky:	Not applicable.
Generic Polymer	Not applicable	
Composite	Not applicable	
Field	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V: at nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py: at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL FLD	Not applicable. Face Loads . Px, Py: local face pressure at nodes.
Body Forces	CBF BFP, BFPE	Constant body forces for element. Xcbf, Ycbf, Ω_x , Ω_y , Ω_z , α_z Body force potentials at nodes/for element. 0, 0,

		0, 0, Xcbf, Ycbf
Velocities	VELO	Velocities. V_x , V_y : at nodes.
Accelerations	ACCE	Accelerations. A_x , A_y : at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. σ_x , σ_θ : axial, circumferential stress. ϵ_x , ϵ_θ : axial, circumferential strain.
	SSIG	Initial stresses/strains at Gauss points. σ_x , σ_θ : axial, circumferential stress. ϵ_x , ϵ_θ : axial, circumferential strain.
Residual Stresses	SSR, SSRE SSRG	Not applicable. Residual stresses at Gauss points. σ_x , σ_θ : axial, circumferential stress.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Stress (default): σ_x , σ_θ : axial, circumferential stress. Strain: ϵ_x , ϵ_θ : axial, circumferential strain.
Modeller	See Results Tables (Appendix K) .

Local Axes

- ☐ [Standard line element](#)

Sign Convention

- ☐ [Standard membrane element](#)

Formulation

Geometric Nonlinearity

Total Lagrangian	For large displacements and small strains.
Updated Lagrangian	Not applicable.
Eulerian	Not applicable.
Co-rotational	Not applicable.

Integration Schemes

Stiffness Default.	1-point (BXM2), 2-point (BXM3).
Fine (see <i>Options</i>).	2-point (BXM2).
Mass Default.	1-point (BXM2), 2-point (BXM3).
Fine (see <i>Options</i>).	2-point (BXM2).

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18** Invokes fine integration rule.
- 36** Follower loads (see *Notes*).
- 47** Use the X-axis as the axis of symmetry.
- 55** Output strains as well as stresses.
- 87** Total Lagrangian geometric nonlinearity.
- 105** Lumped mass matrix.
- 170** Suppress transfer of shape function arrays to disk

Notes on Use

1. The element formulation is based on the standard [isoparametric](#) approach.
2. The variation of stress along the element is constant for BXM2 and linear for BXM3.
3. To apply a non-conservative (follower) pressure load (load type FLD), Option 36 must be specified. Note that this load should be normal to the face and constant for all the nodes of the element. Follower load can only be used with BXM2 elements.
4. The elements should not be used as 'stand-alone' elements if any bending effects are present. The thin axisymmetric shell element BXS3 should be used for this case.
5. The BXM3 element has a zero energy mode which may be excited if the midside node is free and not connected to any other element.
6. When BXM2 elements are used with either variable nodal thicknesses, temperature dependent material properties or utilised in materially nonlinear analyses the 2-point Gauss rule is most effective.

Restrictions

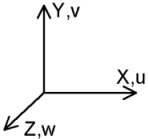
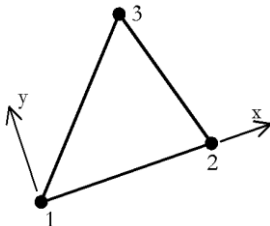
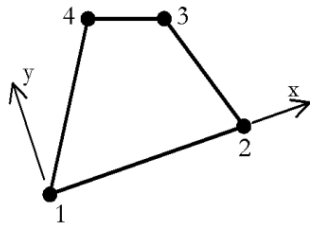
- ☐ [Ensure mid-side node centrality](#)
- ☐ Avoid excessive element curvature
- ☐ Rubber material models can only be used with element BXM2 and must be used with Total Lagrangian geometric nonlinearity (Option 87).

Recommendations on Use

The elements may be used alone to model circular plates or pipes, or coupled with axisymmetric solid elements to provide stiffeners, e.g. radial reinforcement.

3D Space Membrane Elements

General

Element Name	TSM3	SMI4
		
Element Group	Membranes	
Element Subgroup	Space Membranes	
Element Description	A family of space membrane elements in 3D which include a high performance incompatible model (SMI4 only). The elements are intended for 3D membrane structures (they possess no bending stiffness). The elements are formulated in the local element axes which allows directional material properties to be defined relative to the element orientation. The elements can accommodate varying thickness.	
Number Of Nodes	3 or 4 numbered anticlockwise.	
Freedom	U, V, W: at each node.	
Node Coordinates	X, Y, Z: at each node.	

Geometric Properties

t1... tn Thickness at each node.

Material Properties

Linear	Isotropic:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES ORTHOTROPIC (Elastic: Orthotropic Plane Stress)
	Anisotropic:	MATERIAL PROPERTIES ANISOTROPIC 3 (Elastic: Anisotropic Thin Plate)
	Rigidities:	RIGIDITIES 3 (Rigidities: Membrane/Thin Plate)
Matrix	Not applicable	
Joint	Not	

	applicable	
Concrete	Not applicable	
Elasto-Plastic	Not applicable	
Creep	Not applicable	
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable	
Generic Polymer	Not applicable	
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W: at nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py, Pz: at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Uniformly distributed loads. Wx, Wy, Wz: local surface pressures for element.
	FLD	Not applicable.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf, Ω_x , Ω_y , Ω_z , α_x , α_y , α_z
	BFP, BFPE	Body force potentials at nodes/for element. ϕ_1 , ϕ_2 , ϕ_3
Velocities	VELO	Velocities. Vx, Vy, Vz: at nodes.
Accelerations	ACCE	Accelerations. Ax, Ay, Az: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. Nx, Ny, Nxy: forces in local directions. ϵ_x , ϵ_y , γ_{xy} : membrane strains in local directions.
	SSIG	Initial stresses/strains at Gauss points. Nx, Ny, Nxy: forces in local directions. ϵ_x , ϵ_y , γ_{xy} : membrane strains in local directions.

Residual Stresses	Not applicable.	
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

Output

Solver	Stress resultant: N_x , N_y , N_{xy} , N_{max} , N_{min} , β : forces/unit length in local directions. Stress (default): σ_x , σ_y , σ_{xy} , σ_{max} , σ_{min} , β : membrane stresses in local directions. Strain: ϵ_x , ϵ_y , γ_{xy} , ϵ_{max} , ϵ_{min} , β : membrane strains in local directions.
Modeller	See Results Tables (Appendix K) .

Local Axes

- ☐ [Standard area element](#)

Sign Convention

- ☐ [Standard membrane element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Stiffness	Default.	1-point (TSM3), 2x2 (SMI4).
	Fine.	As default.
Mass	Default.	1-point (TSM3), 2x2 (SMI4).
	Fine.	As default.

Mass Modelling

Lumped mass only.

Options

- 32 Suppress stress output but not stress resultants.
- 34 Output stress resultants.
- 55 Output strains as well as stresses.
- 59 Output local direction cosines for elements.
- 77 Output averaged global stresses.

Notes on Use

1. The element formulations are based on the standard
2. The variation of stresses within an element may be regarded as constant for TSM3 and linear for SMI4.
3. The higher performance of SMI4 is due to the addition of 4 incompatible displacement modes.
4. The elements pass the [patch test](#) for mixed triangular and quadrilateral geometry.
5. Distributed loads are lumped at the nodes.
6. The element is formulated so that the material response is evaluated in the local Cartesian system.
7. The SMI4 element is generally the most effective element due to its quadratic displacement accuracy. However, its behaviour tends to deteriorate as the element become distorted.
8. The element matrices are formed using 1-point Gauss quadrature for TSM3. Selective integration is utilised for the evaluation of the element matrices for SMI4. The method used is similar to that proposed by Hughes, with the contribution of the incompatible modes to the strain-displacement matrix being evaluated at the 1-point Gauss rule sampling location and then extrapolated to the 2*2 Gauss rule sampling locations. The element matrices are then formed using the 2*2 Gauss rule.

Restrictions

- ☐ [Avoid excessive aspect ratio.](#)
- ☐ [Avoid excessive warping.](#)

Recommendations on Use

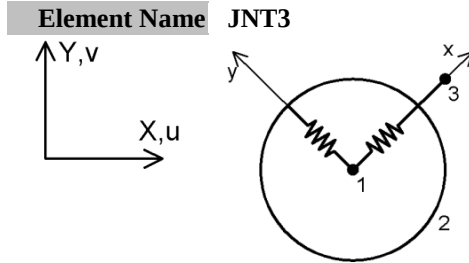
- The space membrane elements have limited 'stand-alone' use because of their inability to support any loading except membrane loading. However, they can be utilised with the flat shell elements (QSI4, TS3) to model very thin membranes in structural components.

- If a structure is composed of exactly co-planar flat space membrane elements that are not stiffened by plate or shell elements, singularities may arise since there is no out-of-plane stiffness.
- If there is a possibility of bending behaviour then a thin shell should be utilised for the analysis.

Chapter 8 : Joint Elements.

2D Joint Element for Bars, Plane Stress and Plane Strain

General



Element Group	Joints
Element Subgroup	2D Joints
Element Description	A 2D joint element which connects two nodes by two springs in the local x and y-directions.
Number Of Nodes	3. The 3rd node is used to define the local x-direction.
Freedom	U, V: at nodes 1 and 2 (active nodes).
Node Coordinates	X, Y: at each node.

Geometric Properties

Not applicable.

Material Properties

Linear	Not applicable	
Matrix	Stiffness:	MATRIX PROPERTIES STIFFNESS 4 K1,..., K10 element stiffness matrix (Not supported in LUSAS Modeller)
	Mass:	MATRIX PROPERTIES MASS 4 M1,..., M10 element mass matrix (Not supported in LUSAS Modeller)
	Damping:	MATRIX PROPERTIES DAMPING 4 C1,..., C10 element damping matrix (Not supported in LUSAS Modeller)
Joint	Standard:	JOINT PROPERTIES 2 (Joint: 2/Spring Stiffness Only)

	Dynamic general:	JOINT PROPERTIES GENERAL 2 (Joint: 2/General Properties)
	Elasto-plastic:	JOINT PROPERTIES NONLINEAR 31 2 (Joint: 2/Elasto-Plastic (Tension and Compression Equal))
	Elasto-plastic:	JOINT PROPERTIES NONLINEAR 32 2 (Joint: 2/Tension and Compression Unequal)
	Nonlinear contact:	JOINT PROPERTIES NONLINEAR 33 2 (Joint: 2/Smooth Contact)
	Nonlinear friction:	JOINT PROPERTIES NONLINEAR 44 2 (Joint: 2/Frictional Contact)
	Viscous damping:	JOINT PROPERTIES NONLINEAR 35 2 (Joint: 2/Viscous Damper)
	Lead-rubber:	JOINT PROPERTIES NONLINEAR 36 2 (Joint: 2/Lead Rubber Bearing)
	Friction pendulum:	JOINT PROPERTIES NONLINEAR 37 2 (Joint: 2/Frictional Pendulum System)
	Multi-linear elastic	JOINT PROPERTIES NONLINEAR 40 2 (Joint: 2/Multi-Linear Elastic)
	Axial force dependent multi-linear elastic	JOINT PROPERTIES NONLINEAR 43 2 (Joint: 2/Axial Force Dependent Multi-Linear Elastic)
Concrete	Not applicable	
Elasto-Plastic	Not applicable	
Creep	Not applicable	
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage	Not applicable	
Rubber	Not applicable	
Generic Polymer	Not applicable	
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V: at active nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py: at active nodes.
Element Loads	Not applicable.	
Distributed Loads	Not applicable.	
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Ω_x , Ω_y , Ω_z , α_z

	BFP, BFPE	Not applicable.
Velocities	VELO	Velocities. V_x , V_y : at nodes.
Accelerations	ACCE	Accelerations. A_x , A_y : at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. F_x , F_y : at active nodes. ϵ_x , ϵ_y : at active nodes.
	SSIG	Not applicable.
Residual Stresses	Not applicable.	
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T_1 , T_2 , T_{10} , T_{20} : actual and initial spring temperatures.
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Force: F_x , F_y : spring forces in local directions. Strain: ϵ_x , ϵ_y : spring strains in local directions.
Modeller	See Results Tables (Appendix K) .

Local Axes

- ☐ [Standard joint element](#)

Sign Convention

- ☐ [Standard joint element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Stiffness	Default.	1-point.
	Fine.	As default.
Mass	Default.	1-point.
	Fine.	As default.

Mass Modelling

Lumped mass only. The position of the mass relative to the two active joint nodes is defined in the joint material data. Point mass elements should be used to model lumped masses when no stiffness modelling is required.

Options

- 55 Output strains as well as stresses.
- 119 Invokes temperature input for joints.

Notes on Use

See [Notes on the use of Joints \(Appendix L\)](#)

Restrictions

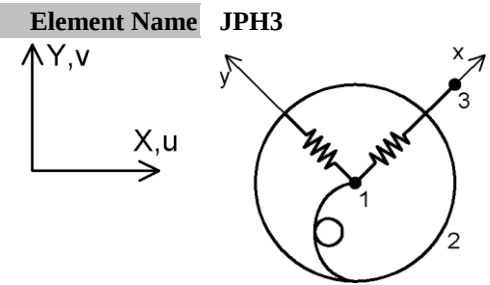
Not applicable.

Recommendations on Use

- The joint elements may be used to release degrees of freedom between elements, e.g. a hinged shell, or to provide nonlinear support conditions, e.g. friction-gap condition. Also, point masses may be represented by including a joint at an element node.
- See [Joint Element Compatibility \(Appendix L\)](#)

2D Joint Element for Engineering and Kirchhoff Beams

General



Element Name	JPH3
Element Group	Joints
Element Subgroup	2D Joints
Element Description	A 2D joint element which connects two nodes by two springs in the local x and y-direction and one spring about the local z-direction.
Number Of Nodes	3. The 3rd node is used to define the local x-direction.
Freedom	U, V, θ_z : at nodes 1 and 2 (active nodes).
Node Coordinates	X, Y: at each node.

Geometric Properties

ey Eccentricity measured from the joint x axis to the nodal line (i.e. parallel to the joint y axis).

Material Properties

Linear Not applicable

Matrix Stiffness:

MATRIX PROPERTIES
STIFFNESS 6 K1,..., K21
element stiffness matrix
(Not supported in LUSAS
Modeller)

Mass:

MATRIX PROPERTIES
MASS 6 M1,..., M21
element mass matrix (Not
supported in LUSAS
Modeller)

Damping:

MATRIX PROPERTIES

		DAMPING 6 C1,..., C21 element damping matrix (Not supported in LUSAS Modeller)
Joint	Standard:	JOINT PROPERTIES 3 (Joint: 3/Spring Stiffness Only)
	Dynamic general:	JOINT PROPERTIES GENERAL 3 (Joint: 3/General Properties)
	Elasto-plastic:	JOINT PROPERTIES NONLINEAR 31 3 (Joint: 3/Elasto-Plastic (Tension and Compression Equal))
	Elasto-plastic:	JOINT PROPERTIES NONLINEAR 32 3 (Joint: 3/Tension and Compression Unequal)
	Nonlinear contact:	JOINT PROPERTIES NONLINEAR 33 3 (Joint: 3/Smooth Contact)
	Nonlinear friction:	JOINT PROPERTIES NONLINEAR 44 3 (Joint: 3/Frictional Contact)
	Viscous damping:	JOINT PROPERTIES NONLINEAR 35 3 (Joint: 3/Viscous Damper)
	Lead-rubber:	JOINT PROPERTIES NONLINEAR 36 3 (Joint: 3/Lead Rubber Bearing)
	Friction pendulum:	JOINT PROPERTIES NONLINEAR 37 3 (Joint: 3/Frictional Pendulum System)
	Multi-linear elastic	JOINT PROPERTIES NONLINEAR 40 3 (Joint: 3/Multi-Linear Elastic)
	Axial force dependent multi-linear elastic	JOINT PROPERTIES NONLINEAR 43 3 (Joint: 3/Axial Force Dependent Multi-Linear Elastic)
Joint	Standard:	JOINT PROPERTIES 3 (Joint: 3/Spring Stiffness Only)
	Dynamic general:	JOINT PROPERTIES

GENERAL 3 (Joint:
3/General Properties)

Concrete Not applicable
Elasto-Plastic Not applicable
Creep Not applicable
Damage Not applicable
Viscoelastic Not applicable
Shrinkage Not applicable

Rubber Not applicable
Generic Polymer Not applicable
Composite Not applicable

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, θ_z : at active nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py, Mz: at active nodes.
Element Loads	Not applicable	
Distributed Loads	Not applicable	
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, $\Omega_x, \Omega_y, \Omega_z, \alpha_z$
	BFP, BFPE	Not applicable.
Velocities	VELO	Velocities. Vx, Vy: at nodes.
Accelerations	ACCE	Accelerations. Ax, Ay: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. Resultants. Fx, Fy, Mz: spring forces and moment in local directions. $\epsilon_x, \epsilon_y, \psi_z$: strains at nodes.
	SSIG	Not applicable.
Residual Stresses	Not applicable	
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T1, T2, T3, T10, T20, T30: actual and initial spring temperatures.
Field Loads	Not applicable	

Temp Not applicable
Dependent
Loads

LUSAS Output

Solver Force: F_x , F_y , M_z : spring forces and moment in local directions.
Strain: ϵ_x , ϵ_y , ψ_z : spring strains in local directions.
Modeller See [Results Tables \(Appendix K\)](#).

Local Axes

- ☐ [Standard joint element](#)

Sign Convention

- ☐ [Standard joint element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Stiffness Default. 1-point.
Fine. As default.
Mass Default. 1-point.
Fine. As default.

Mass Modelling

Lumped mass only. The position of the mass relative to the two active joint nodes is defined in the joint material data. Point mass elements should be used to model lumped masses when no stiffness modelling is required.

Options

- 55 Output strains as well as stresses.
- 119 Invokes temperature input for joints.

Notes on Use

See [Notes on the use of Joints \(Appendix L\)](#)

Restrictions

Not applicable.

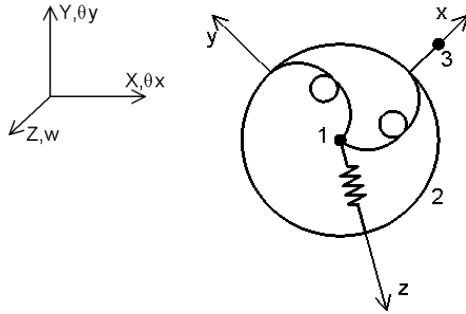
Recommendations on Use

- The joint elements may be used to release degrees of freedom between elements, e.g. a hinged shell, or to provide nonlinear support conditions, e.g. friction-gap condition. Also, point masses may be represented by including a joint at an element node.
- See [Joint Element Compatibility \(Appendix L\)](#)

2D Joint Element for Grillage Beams and Plates

General

Element Name JF3



Element Group	Joints
Element Subgroup	2D Joints
Element Description	A 2D joint element which connects two nodes by one spring in the local z-direction and two springs about the x and y directions.
Number Of Nodes	3. The 3rd node is used to define the local x-direction.
Freedom	W, θ_x, θ_y : at nodes 1 and 2 (active nodes).
Node Coordinates	X, Y : at each node.

Geometric Properties

Not applicable.

Material Properties

Linear Not applicable

Matrix	Stiffness:	MATRIX PROPERTIES STIFFNESS 6 $K_1, \dots, K_{21}$ element stiffness matrix (Not supported in LUSAS Modeller)
	Mass:	MATRIX PROPERTIES MASS 6 $M_1, \dots, M_{21}$ element mass matrix (Not supported in LUSAS Modeller)
	Damping:	MATRIX PROPERTIES DAMPING 6 $C_1, \dots, C_{21}$ element damping matrix (Not supported in LUSAS Modeller)
Joint	Standard:	JOINT PROPERTIES 3 (Joint: 3/Spring)

		Stiffness Only)
	Dynamic general:	JOINT PROPERTIES GENERAL 3 (Joint: 3/General Properties)
	Elasto-plastic:	JOINT PROPERTIES NONLINEAR 31 3 (Joint: 3/Elasto-Plastic (Tension and Compression Equal))
	Elasto-plastic:	JOINT PROPERTIES NONLINEAR 32 3 (Joint: 3/Tension and Compression Unequal)
	Nonlinear contact:	JOINT PROPERTIES NONLINEAR 33 3 (Joint: 3/Smooth Contact)
	Nonlinear friction:	Not applicable
	Viscous damping:	JOINT PROPERTIES NONLINEAR 35 3 (Joint: 3/Viscous Damper)
	Lead-rubber:	Not applicable
	Friction pendulum:	Not applicable
	Multi-linear elastic	JOINT PROPERTIES NONLINEAR 40 3 (Joint: 3/Multi-Linear Elastic)
	Axial force dependent multi-linear elastic	JOINT PROPERTIES NONLINEAR 43 3 (Joint: 3/Axial Force Dependent Multi-Linear Elastic)
Concrete	Not applicable	
Elasto-Plastic	Not applicable	
Creep	Not applicable	
Damage	Not applicable	
Viscoelastic	Not applicable.	
Shrinkage	Not applicable	
Rubber	Not applicable	
Generic Ploymer	Not applicable	
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. ω , θ_x , θ_y : at active nodes.
Concentrated Loads	CL	Concentrated loads. P_z , M_x , M_y : at active nodes.
Element Loads	Not applicable	
Distributed Loads	Not applicable	

Body Forces	CBF	Constant body forces for element. Zcbf
	BFP, BFPE	Not applicable.
Velocities	VELO	Velocities. Vz: at nodes.
Accelerations	ACCE	Accelerations. Az: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. Fz, Mx, My: at active nodes. ϵ_z , ψ_x , ψ_y : at active nodes.
	SSIG	Not applicable.
Residual Stresses	Not applicable	
Temperatures	TEMP,	Temperatures at nodes/for element. T ₁ , T ₂ , T ₃ , T ₁₀ , T ₂₀ , T ₃₀ : actual and initial spring temperatures.
	TMPE	
Field Loads	Not applicable	
Temp Dependent Loads	Not applicable	

LUSAS Output

Solver	Force: Pz, Mx, My: spring forces in local directions. Strain: ϵ_z , ψ_x , ψ_y : spring strains in local directions.
Modeller	See Results Tables (Appendix K) .

Local Axes

- ☐ [Standard joint element](#)

Sign Convention

- ☐ [Standard joint element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Stiffness	Default.	1-point.
	Fine.	As default.
Mass	Default.	1-point.
	Fine.	As default.

Mass Modelling

Lumped mass only. The position of the mass relative to the two active joint nodes is defined in the joint material data. Point mass elements should be used to model lumped masses when no stiffness modelling is required.

Options

- 55 Output strains as well as stresses.
- 119 Invokes temperature input for joints.

Notes on Use

See [Notes on the use of Joints \(Appendix L\)](#)

Restrictions

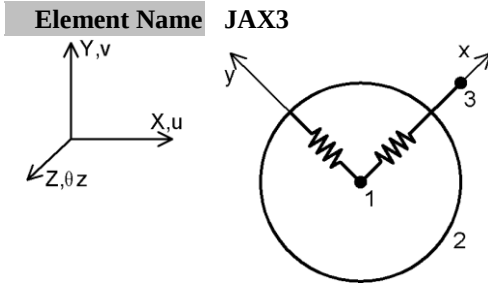
Not applicable.

Recommendations on Use

- The joint elements may be used to release degrees of freedom between elements, e.g. a hinged shell, or to provide nonlinear support conditions, e.g. friction-gap condition. Also, point masses may be represented by including a joint at an element node.
- See [Joint Element Compatibility \(Appendix L\)](#)

2D Joint Element for Axisymmetric Solids

General



Element Group	Joints
Element Subgroup	2D Joints
Element Description	An axisymmetric joint element for use with axisymmetric solid elements, which connects two nodes by two springs in the local x and y-directions and a 3rd spring in the circumferential direction.
Number Of Nodes	3. The 3rd node is used to define the local x-direction.
Freedom Node Coordinates	U, V: at nodes 1 and 2 (active nodes). X, Y: at each node.

Geometric Properties

Not applicable.

Material Properties

Linear	Not applicable
Matrix	Stiffness: MATRIX PROPERTIES STIFFNESS 6 K1,..., K10 element stiffness matrix (Not supported in LUSAS Modeller)
	Mass: MATRIX PROPERTIES MASS 6 M1,..., M10 element mass matrix (Not supported in LUSAS Modeller)
	Damping: MATRIX PROPERTIES DAMPING 6 C1,..., C10 element damping matrix (Not supported in LUSAS Modeller)
Joint	Standard: JOINT PROPERTIES 2 (Joint: 2/Spring Stiffness Only) (See notes on use)
	Dynamic general: JOINT PROPERTIES GENERAL 2 (Joint:

		2/General Properties) (See notes on use)
Elasto-plastic:		JOINT PROPERTIES NONLINEAR 31 2 (Joint: 2/Elasto-Plastic (Tension and Compression Equal)) (See notes on use)
Elasto-plastic:		JOINT PROPERTIES NONLINEAR 32 2 (Joint: 2/Tension and Compression Unequal) (See notes on use)
Nonlinear contact:		JOINT PROPERTIES NONLINEAR 33 2 (Joint: 2/Smooth Contact) (See notes on use)
Nonlinear friction:		JOINT PROPERTIES NONLINEAR 44 2 (Joint: 2/Frictional Contact) (See notes on use)
Viscous damping:		JOINT PROPERTIES NONLINEAR 35 2 (Joint: 2/Viscous Damper) (See notes on use)
Lead-rubber:		JOINT PROPERTIES NONLINEAR 36 2 (Joint: 2/Lead Rubber Bearing) (See notes on use)
Friction pendulum:		JOINT PROPERTIES NONLINEAR 37 2 (Joint: 2/Frictional Pendulum System) (See notes on use)
Multi-linear elastic		JOINT PROPERTIES NONLINEAR 40 2 (Joint: 2/Multi-Linear Elastic)
Axial force dependent multi- linear elastic		JOINT PROPERTIES NONLINEAR 43 2 (Joint: 2/Axial Force Dependent Multi-Linear Elastic)
Concrete	Not applicable	
Elasto- Plastic	Not applicable	
Creep	Not applicable	
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage	Not applicable	
Rubber	Not applicable	
Generic	Not applicable	
Polymer		
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V: at active nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py: at active nodes.
Element Loads	Not applicable.	
Distributed	Not applicable.	

Loads		
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Ω_x , Ω_y , Ω_z , α_z
	BFP, BFPE	Not applicable.
Velocities	VELO	Velocities. Vx, Vy: at nodes.
Accelerations	ACCE	Accelerations. Ax, Ay: at nodes..
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. Fx, Fy: spring forces in local directions. ϵ_x , ϵ_y : spring strains in local directions.
	SSIG	Not applicable.
Residual Stresses	Not applicable.	
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T1, T2, T10, T20: actual and initial spring temperatures.
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Force: Fx, Fy, Fz: spring forces in local directions. Strain: ϵ_x , ϵ_y , ϵ_z : spring strains in local directions.
Modeller	See Results Tables (Appendix K) .

Local Axes

- ☐ [Standard joint element](#)

Sign Convention

- ☐ [Standard joint element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Stiffness	Default.	1-point.
	Fine.	As default.

Mass	Default.	1-point.
	Fine.	As default.

Mass Modelling

Lumped mass only. The position of the mass relative to the two active joint nodes is defined in the joint material data. Point mass elements should be used to model lumped masses when no stiffness modelling is required.

Options

- 47 X-axis taken as axis of symmetry.
- 55 Output strains as well as stresses.
- 119 Invokes temperature input for joints.

Notes on Use

- For general notes see [Notes on the use of Joints \(Appendix L\)](#)
- This joint has only two degrees of freedom but requires 3 inputs. The 3rd input required is the circumferential stiffness.
- For problems where the circumferential forces are to be transmitted by adjacent elements the circumferential stiffness should be input as zero.
- This element cannot be used with axisymmetric Fourier elements.

Restrictions

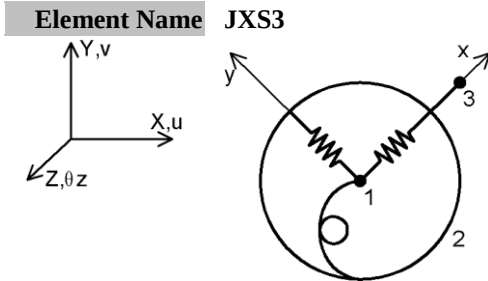
Not applicable.

Recommendations on Use

- The joint elements may be used to release degrees of freedom between elements, e.g. a hinged shell, or to provide nonlinear support conditions, e.g. friction-gap condition. Also, point masses may be represented by including a joint at an element node.
- See [Joint Element Compatibility \(Appendix L\)](#)

2D Joint Element for Axisymmetric Shells

General



Element Group	Joints
Element Subgroup	2D Joints
Element Description	An axisymmetric joint element for use with axisymmetric shell elements, which connects two nodes by two springs in the local x and y-directions, one spring about the local z-direction and a 4th spring in the circumferential direction.
Number Of Nodes	3. The 3rd node is used to define the local x-direction.
Freeds	U, V, θ : at nodes 1 and 2 (active nodes).
Node Coordinates	X, Y: at each node.

Geometric Properties

Not applicable.

Material Properties

Linear	Not applicable
Matrix	Stiffness: MATRIX PROPERTIES STIFFNESS 8 K1,..., K21 element stiffness matrix (Not supported in LUSAS Modeller)
	Mass: MATRIX PROPERTIES MASS 8 M1,..., M21 element mass matrix (Not supported in LUSAS Modeller)
	Damping: MATRIX PROPERTIES DAMPING 8 C1,..., C21 element damping matrix (Not supported in LUSAS Modeller)
Joint	Standard: JOINT PROPERTIES 3 (Joint: 3/Spring)

		Stiffness Only) (See notes on use)
Dynamic general:		JOINT PROPERTIES GENERAL 3 (Joint: 3/General Properties) (See notes on use)
Elasto-plastic:		JOINT PROPERTIES NONLINEAR 31 3 (Joint: 3/Elasto-Plastic (Tension and Compression Equal)) (See notes on use)
Elasto-plastic:		JOINT PROPERTIES NONLINEAR 32 3 (Joint: 3/Tension and Compression Unequal) (See notes on use)
Nonlinear contact:		JOINT PROPERTIES NONLINEAR 33 3 (Joint: 3/Smooth Contact) (See notes on use)
Nonlinear friction:		JOINT PROPERTIES NONLINEAR 44 3 (Joint: 3/Frictional Contact) (See notes on use)
Viscous damping:		JOINT PROPERTIES NONLINEAR 35 3 (Joint: 3/Viscous Damper) (See notes on use)
Lead-rubber:		JOINT PROPERTIES NONLINEAR 36 3 (Joint: 3/Lead Rubber Bearing) (See notes on use)
Friction pendulum:		JOINT PROPERTIES NONLINEAR 37 3 (Joint: 3/Frictional Pendulum System) (See notes on use)
Multi-linear elastic		JOINT PROPERTIES NONLINEAR 40 3 (Joint: 3/Multi-Linear Elastic)
Axial force dependent multi-linear elastic		JOINT PROPERTIES NONLINEAR 43 3 (Joint: 3/Axial Force Dependent Multi-Linear Elastic)
Concrete	Not applicable	
Elasto-Plastic	Not applicable	
Creep	Not applicable	
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage	Not applicable	
Rubber	Not applicable	
Generic	Not applicable	
Polymer		
Composite	Not applicable	

Loading

Prescribed Value PDSP, TPDSP Prescribed variable. U, V, θ : at active nodes.

Concentrated Loads	CL	Concentrated loads. Px, Py, M: at active nodes.
Element Loads	Not applicable.	
Distributed Loads	Not applicable.	
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Ω_x , Ω_y , Ω_z , α_z
	BFP, BFPE	Not applicable.
Velocities	VELO	Velocities. Vx, Vy: at nodes.
Accelerations	ACCE	Accelerations. Ax, Ay: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. Fx, Fy: spring forces in local directions. ϵ_x , ϵ_y : spring strains in local directions.
	SSIG	Not applicable.
Residual Stresses	Not applicable.	
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T1, T2, T3, T10, T20, T30: actual and initial spring temperatures.
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Force: Fx, Fy, Fz, M: spring forces in local directions. Strain: ϵ_x , ϵ_y , ϵ_z , ψ_z : spring strains in local directions.
Modeller	See Results Tables (Appendix K) .

Local Axes

- ☐ [Standard joint element](#)

Sign Convention

- ☐ [Standard joint element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Stiffness	Default.	1-point.
	Fine.	As default.
Mass	Default.	1-point.
	Fine.	As default.

Mass Modelling

Lumped mass only. The position of the mass relative to the two active joint nodes is defined in the joint material data. Point mass elements should be used to model lumped masses when no stiffness modelling is required.

Options

- 47 X-axis taken as axis of symmetry.
- 55 Output strains as well as stresses.
- 119 Invokes temperature input for joints.

Notes on Use

This joint has only three degrees of freedom but requires 4 inputs. The 4th input required is the circumferential stiffness.

For general notes see [Notes on the use of Joints \(Appendix L\)](#)

Restrictions

Not applicable.

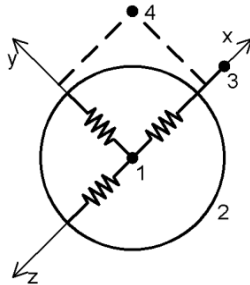
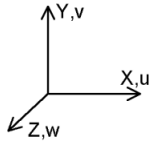
Recommendations on Use

- The joint elements may be used to release degrees of freedom between elements, e.g. a hinged shell, or to provide nonlinear support conditions, e.g. friction-gap condition. Also, point masses may be represented by including a joint at an element node.
- See [Joint Element Compatibility \(Appendix L\)](#)

3D Joints for Bars, Solids and Space Membranes

General

Element Name JNT4



Element Group Joints

Element 3D Joints

Subgroup

Element Description A 3D joint element which connects two nodes by three springs in the local x, y and z-directions.

Number Of Nodes 4. The 3rd and 4th nodes are used to define the local x-axis and local xy-plane.

Freedom Node U, V, W: at nodes 1 and 2 (active nodes).

Coordinates X, Y, Z: at each node.

Geometric Properties

Not applicable.

Material Properties

Linear Not applicable

Matrix Stiffness: MATRIX PROPERTIES STIFFNESS 6 K1,..., K21 element stiffness matrix (Not supported in LUSAS Modeller)

Mass: MATRIX PROPERTIES MASS 6 M1,..., M21 element mass matrix (Not supported in LUSAS Modeller)

Damping: MATRIX PROPERTIES DAMPING 6 C1,..., C21 element damping matrix (Not supported in LUSAS Modeller)

Joint Standard: JOINT PROPERTIES 3 (Joint: 3/Spring Stiffness Only)

Dynamic general:	JOINT PROPERTIES GENERAL 3 (Joint: 3/General Properties)
Elasto-plastic:	JOINT PROPERTIES NONLINEAR 31 3 (Joint: 3/Elasto-Plastic (Tension and Compression Equal))
Elasto-plastic:	JOINT PROPERTIES NONLINEAR 32 3 (Joint: 3/Tension and Compression Unequal)
Nonlinear contact:	JOINT PROPERTIES NONLINEAR 33 3 (Joint: 3/Smooth Contact)
Nonlinear friction:	JOINT PROPERTIES NONLINEAR 44 3 (Joint: 3/Frictional Contact)
Viscous damping:	JOINT PROPERTIES NONLINEAR 35 3 (Joint: 3/Viscous Damper)
Lead-rubber:	JOINT PROPERTIES NONLINEAR 36 3 (Joint: 3/Lead Rubber Bearing)
Friction pendulum:	JOINT PROPERTIES NONLINEAR 37 3 (Joint: 3/Frictional Pendulum System)
Multi-linear elastic	JOINT PROPERTIES NONLINEAR 40 3 (Joint: 3/Multi-Linear Elastic)
Axial force dependent multi- linear elastic	JOINT PROPERTIES NONLINEAR 43 3 (Joint: 3/Axial Force Dependent Multi-Linear Elastic)
Concrete	Not applicable
Elasto-Plastic	Not applicable
Creep	Not applicable
Damage	Not applicable
Viscoelastic	Not applicable
Shrinkage	Not applicable
Rubber	Not applicable
Generic Polymer	Not applicable
Composite	Not applicable

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W: at active nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py, Pz: at active nodes.
Element Loads	Not applicable.	
Distributed Loads	Not applicable.	

Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf, Ω_x , Ω_y , Ω_z , α_x , α_y , α_z
	BFP, BFPE	Not applicable.
Velocities	VELO	Velocities. Vx, Vy, Vz: at nodes.
Accelerations	ACCE	Accelerations. Ax, Ay, Az: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. Fx, Fy, Fz: spring forces in local directions. ϵ_x , ϵ_y , ψ_z : spring strains in local directions.
	SSIG	Not applicable.
	Residual Stresses	Not applicable.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T1, T2, T3, T10, T20, T30: actual and initial spring temperatures.
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Force: Fx, Fy, Fz: spring forces in local directions. Strain: ϵ_x , ϵ_y , ϵ_z : spring strains in local directions.
Modeller	See Results Tables (Appendix K) .

Local Axes

- ❑ [Standard joint element](#)

Sign Convention

- ❑ [Standard joint element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Stiffness	Default.	1-point.
	Fine.	As default.

Mass Default.	1-point.
Fine	As default.

Mass Modelling

Lumped mass only. The position of the mass relative to the two active joint nodes is defined in the joint material data. Point mass elements should be used to model lumped masses when no stiffness modelling is required.

Options

- 55 Output strains as well as stresses.
- 119 Invokes temperature input for joints.

Notes on Use

- For general notes see [Notes on the use of Joints \(Appendix L\)](#)

Restrictions

Not applicable.

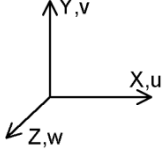
Recommendations on Use

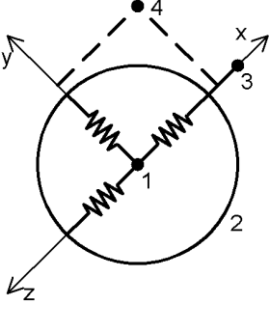
- The joint elements may be used to release degrees of freedom between elements, e.g. a hinged shell, or to provide nonlinear support conditions, e.g. friction-gap condition. Also, point masses may be represented by including a joint at an element node.
- See [Joint Element Compatibility \(Appendix L\)](#)

3D Joints for Semiloof Shells

General

Element Name	JL43





Element Group	Joints
Element Subgroup	3D Joints
Element Description	A 3D joint element which connects two nodes by three springs in the local x, y and z-directions.
Number Of Nodes	4. The 3rd and 4th nodes are used to define the local x-axis and local xy-plane.
Freedom	U, V, W: at nodes 1 and 2 (active nodes).
Node Coordinates	X, Y, Z: at each node.

Geometric Properties

Not applicable.

Material Properties

Linear	Not applicable
Matrix	Stiffness: MATRIX PROPERTIES STIFFNESS 6 K1,..., K21 element stiffness matrix (Not supported in LUSAS Modeller)
	Mass: MATRIX PROPERTIES MASS 6 M1,..., M21 element mass matrix (Not supported in LUSAS Modeller)
	Damping: MATRIX PROPERTIES DAMPING 6 C1,..., C21 element damping matrix (Not supported in LUSAS Modeller)
Joint	Standard: JOINT PROPERTIES 3 (Joint: 3/Spring Stiffness Only)

Dynamic general:	JOINT PROPERTIES GENERAL 3 (Joint: 3/General Properties)
Elasto-plastic:	JOINT PROPERTIES NONLINEAR 31 3 (Joint: 3/Elasto-Plastic (Tension and Compression Equal))
Elasto-plastic:	JOINT PROPERTIES NONLINEAR 32 3 (Joint: 3/Tension and Compression Unequal)
Nonlinear contact:	JOINT PROPERTIES NONLINEAR 33 3 (Joint: 3/Smooth Contact)
Nonlinear friction:	JOINT PROPERTIES NONLINEAR 44 3 (Joint: 3/Frictional Contact)
Viscous damping:	JOINT PROPERTIES NONLINEAR 35 3 (Joint: 3/Viscous Damper)
Lead-rubber:	JOINT PROPERTIES NONLINEAR 36 3 (Joint: 3/Lead Rubber Bearing)
Friction pendulum:	JOINT PROPERTIES NONLINEAR 37 3 (Joint: 3/Frictional Pendulum System)
Multi-linear elastic	JOINT PROPERTIES NONLINEAR 40 3 (Joint: 3/Multi-Linear Elastic)
Axial force dependent multi- linear elastic	JOINT PROPERTIES NONLINEAR 43 3 (Joint: 3/Axial Force Dependent Multi-Linear Elastic)
Concrete	Not applicable
Elasto-Plastic	Not applicable
Creep	Not applicable
Damage	Not applicable
Viscoelastic	Not applicable
Shrinkage	Not applicable
Rubber	Not applicable
Generic Polymer	Not applicable
Composite	Not applicable

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W: at active nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py, Pz: at active nodes.
Element Loads	Not applicable.	
Distributed Loads	Not applicable.	

Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf, Ω_x , Ω_y , Ω_z , α_x , α_y , α_z
	BFP, BFPE	Not applicable.
Velocities	VELO	Velocities. Vx, Vy, Vz: at nodes.
Accelerations	ACCE	Accelerations. Ax, Ay, Az: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. Fx, Fy, Fz: spring forces in local directions. ϵ_x , ϵ_y , ψ_z : spring strains in local directions.
	SSIG	Not applicable.
	Residual Stresses	Not applicable.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T1, T2, T3, T10, T20, T30: actual and initial spring temperatures.
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Force: Fx, Fy, Fz: spring forces in local directions. Strain: ϵ_x , ϵ_y , ϵ_z : spring strains in local directions.
Modeller	See Results Tables (Appendix K) .

Local Axes

- ❑ [Standard joint element](#)

Sign Convention

- ❑ [Standard joint element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Stiffness	Default.	1-point.
	Fine.	As default.

Mass	Default.	1-point.
	Fine.	As default.

Mass Modelling

Lumped mass only. The position of the mass relative to the two active joint nodes is defined in the joint material data. Point mass elements should be used to model lumped masses when no stiffness modelling is required.

Options

- 55 Output strains as well as stresses.
- 119 Invokes temperature input for joints.

Notes on Use

- For general notes see [Notes on the use of Joints \(Appendix L\)](#)
- When using Modeller to assign this semiloof joint element to interface lines a JL43 joint element will be created at the semiloof shell corner nodes and a JSL4 joint element will be created at the semiloof shell mid-side nodes.

Restrictions

Not applicable.

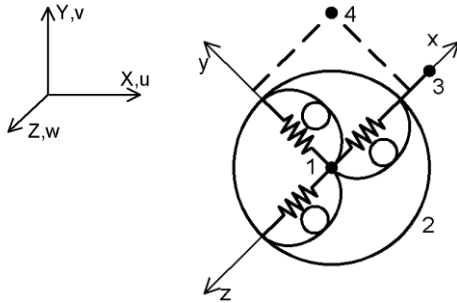
Recommendations on Use

- The joint elements may be used to release degrees of freedom between elements, e.g. a hinged shell, or to provide nonlinear support conditions, e.g. friction-gap condition. Also, point masses may be represented by including a joint at an element node.
- See [Joint Element Compatibility \(Appendix L\)](#)

3D Joint Elements for Engineering, Kirchhoff and Semiloof Beams

General

Element Name JSH4, JL46



Element Group Joints

Element 3D Joints

Subgroup

Element Description 3D joint elements which connects two nodes by six springs in the local x, y and z-directions. Use JL46 for semiloof beam end nodes. 4. The 3rd and 4th nodes are used to define the local x-axis and local xy-plane respectively.

Number Of Nodes 4. The 3rd and 4th nodes are used to define the local x-axis and local xy-plane respectively.

Freedoms U, V, W, θ_x , θ_y , θ_z : at nodes 1 and 2 (active nodes).

Node Coordinates X, Y, Z: at each node.

Geometric Properties

ez Eccentricity measured from the joint xy-plane to the nodal line.

Material Properties

Linear Not applicable

Matrix Stiffness:

MATRIX PROPERTIES STIFFNESS 12 K1,..., K78 element stiffness matrix (Not supported in LUSAS Modeller)

Mass:

MATRIX PROPERTIES MASS 12 M1,..., M78 element mass matrix (Not supported in LUSAS Modeller)

Damping:

MATRIX PROPERTIES DAMPING 12 C1,..., C78 element damping matrix (Not supported in LUSAS Modeller)

Joint	Standard:	JOINT PROPERTIES 6 (Joint: 6/Spring Stiffness Only)
	Dynamic general:	JOINT PROPERTIES GENERAL 6 (Joint: 6/General Properties)
	Elasto-plastic:	JOINT PROPERTIES NONLINEAR 31 6 (Joint: 6/Elasto-Plastic (Tension and Compression Equal))
	Elasto-plastic:	JOINT PROPERTIES NONLINEAR 32 6 (Joint: 6/Tension and Compression Unequal)
	Nonlinear contact:	JOINT PROPERTIES NONLINEAR 33 6 (Joint: 6/Smooth Contact)
	Nonlinear friction:	JOINT PROPERTIES NONLINEAR 44 6 (Joint: 6/Frictional Contact)
	Viscous damping:	JOINT PROPERTIES NONLINEAR 35 6 (Joint: 6/Viscous Damper)
	Lead-rubber:	JOINT PROPERTIES NONLINEAR 36 6 (Joint: 6/Lead Rubber Bearing)
	Friction pendulum:	JOINT PROPERTIES NONLINEAR 37 6 (Joint: 6/Frictional Pendulum System)
	Multi-linear elastic	JOINT PROPERTIES NONLINEAR 40 6 (Joint: 6/Multi-Linear Elastic)
	Axial force dependent multi-linear elastic	JOINT PROPERTIES NONLINEAR 43 6 (Joint: 6/Axial Force Dependent Multi-Linear Elastic)
Concrete	Not applicable	
Elasto-Plastic	Not applicable	
Creep	Not applicable	
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage	Not applicable	
Rubber	Not applicable	
Generic Polymer	Not applicable	
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W, θ_x , θ_y , θ_z : at active nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py, Pz, Mx, My, Mz: at active nodes.
Element Loads	Not	

		applicable.
Distributed Loads	Not applicable.	
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf, Ω_x , Ω_y , Ω_z , α_x , α_y , α_z
	BFP, BFPE	Not applicable.
Velocities	VELO	Velocities. Vx, Vy, Vz: at nodes.
Accelerations	ACCE	Accelerations. Ax, Ay, Az: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. Fx, Fy, Fz, Mx, My, Mz: spring forces in local directions. ϵ_x , ϵ_y , ϵ_z , ψ_x , ψ_y , ψ_z : spring strains in local directions.
	SSIG	Not applicable.
Residual Stresses	Not applicable.	
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T1, T2, T3, T4, T5, T6, T10, T20, T30, T40, T50, T60: actual and initial spring temperatures.
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Force: Fx, Fy, Fz, Mx, My, Mz spring forces in local directions. Strain: ϵ_x , ϵ_y , ϵ_z , ψ_x , ψ_y , ψ_z : spring strains in local directions.
Modeller	See Results Tables (Appendix K) .

Local Axes

- ☐ [Standard joint element](#)

Sign Convention

- ☐ [Standard joint element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Stiffness Default.	1-point.
Fine.	As default.
Mass Default.	1-point.
Fine	As default.

Mass Modelling

Lumped mass only. The position of the mass relative to the two active joint nodes is defined in the joint material data. Point mass elements should be used to model lumped masses when no stiffness modelling is required.

Options

- 55 Output strains as well as stresses.
- 119 Invokes temperature input for joints

Notes on Use

See [Notes on the use of Joints \(Appendix L\)](#)

Restrictions

Not applicable.

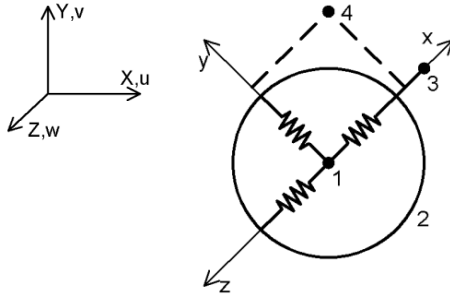
Recommendations on Use

- The joint elements may be used to release degrees of freedom between elements, e.g. a hinged shell, or to provide nonlinear support conditions, e.g. friction-gap condition. Also, point masses may be represented by including a joint at an element node.
- See [Joint Element Compatibility \(Appendix L\)](#)

3D Joint Element for Semiloof Beams

General

Element Name JSL4



Element Group Joints

Element Subgroup 3D Joints

Description A 3D joint element which connects two nodes by three springs in the local x, y and z-directions and two springs about the local x-direction at the 1st and 2nd loof points.

Number Of Nodes 4. The 3rd and 4th nodes are used to define the local x-axis and local xy-plane respectively.

Freedoms U, V, W, θ_1 , θ_2 : at nodes 1 and 2 (active nodes).

Node Coordinates X, Y, Z: at each node.

Geometric Properties

Not applicable.

Material Properties

Linear Not applicable

Matrix Stiffness: MATRIX PROPERTIES STIFFNESS 10 K1,..., K55 element stiffness matrix (Not supported in LUSAS Modeller)

Mass: MATRIX PROPERTIES MASS 10 M1,..., M55 element mass matrix (Not supported in LUSAS Modeller)

Damping: MATRIX PROPERTIES DAMPING 10 C1,..., C55 element damping matrix (Not supported in LUSAS Modeller)

Joint Standard: JOINT PROPERTIES 5 (Joint: 5/Spring

		Stiffness Only)
Dynamic general:		JOINT PROPERTIES GENERAL 5 (Joint: 5/General Properties)
Elasto-plastic:		JOINT PROPERTIES NONLINEAR 31 5 (Joint: 5/Elasto-Plastic (Tension and Compression Equal))
Elasto-plastic:		JOINT PROPERTIES NONLINEAR 32 5 (Joint: 5/Tension and Compression Unequal)
Nonlinear contact:		JOINT PROPERTIES NONLINEAR 33 5 (Joint: 5/Smooth Contact)
Nonlinear friction:		JOINT PROPERTIES NONLINEAR 44 5 (Joint: 5/Frictional Contact)
Viscous damping:		JOINT PROPERTIES NONLINEAR 35 5 (Joint: 5/Viscous Damper)
Lead-rubber:		JOINT PROPERTIES NONLINEAR 36 5 (Joint: 5/Lead Rubber Bearing)
Friction pendulum:		JOINT PROPERTIES NONLINEAR 37 5 (Joint: 5/Frictional Pendulum System)
Multi-linear elastic		JOINT PROPERTIES NONLINEAR 40 5 (Joint: 5/Multi-Linear Elastic)
Axial force dependent multi- linear elastic		JOINT PROPERTIES NONLINEAR 43 5 (Joint: 5/Axial Force Dependent Multi-Linear Elastic)
Concrete	Not applicable	
Elasto-Plastic	Not applicable	
Creep	Not applicable	
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage	Not applicable	
Rubber	Not applicable	
Generic Polymer	Not applicable	
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W, θ_1 , θ_2 : at active nodes.
Concentrated Loads	CL	Concentrated loads. Px, Py, Pz, M1, M2: at active nodes.
Element Loads	Not applicable.	

Distributed Loads	Not applicable.	
Body Forces	CBF	Constant body forces for element. X_{cbf} , Y_{cbf} , Z_{cbf} , Ω_x , Ω_y , Ω_z , α_x , α_y , α_z
	BFP, BFPE	Not applicable.
Velocities	VELO	Velocities. V_x , V_y , V_z : at nodes.
Accelerations	ACCE	Accelerations. A_x , A_y , A_z : at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. F_x , F_y , F_z , M_x , M_y , M_z : spring forces in local directions. ϵ_x , ϵ_y , ϵ_z , ψ_x , ψ_y , ψ_z : spring strains in local directions.
	SSIG	Not applicable.
Residual Stresses	Not applicable.	
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T_1 , T_2 , T_3 , T_4 , T_5 , T_{10} , T_{20} , T_{30} , T_{40} , T_{50} : actual and initial spring temperatures.
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Force: F_x , F_y , F_z , M_1 , M_2 : spring forces in local directions. Strain: ϵ_x , ϵ_y , ϵ_z , ψ_1 , ψ_2 : spring strains in local directions.
Modeller	See Results Tables (Appendix K) .

Local Axes

- ☐ [Standard joint element](#)

Sign Convention

- ☐ [Standard joint element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Stiffness	Default.	1-point.
	Fine.	As default.
Mass	Default.	1-point.
	Fine.	As default.

Mass Modelling

Lumped mass only. The position of the mass relative to the two active joint nodes is defined in the joint material data. Point mass elements should be used to model lumped masses when no stiffness modelling is required.

Options

- 55 Output strains as well as stresses.
- 119 Invokes temperature input for joints.

Notes on Use

See [Notes on the use of Joints \(Appendix L\)](#)

Restrictions

Not applicable.

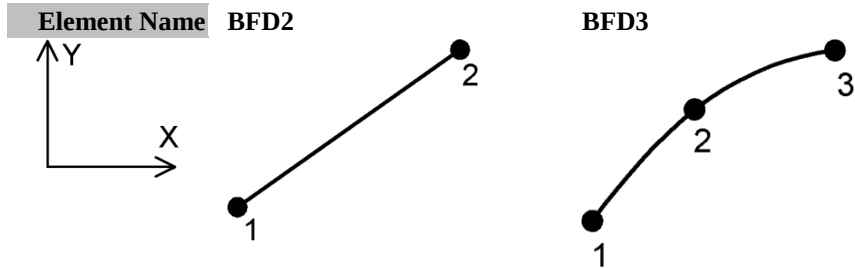
Recommendations on Use

- The joint elements may be used to release degrees of freedom between elements, e.g. a hinged shell, or to provide nonlinear support conditions, e.g. friction-gap condition. Also, point masses may be represented by including a joint at an element node.
- See [Joint Element Compatibility \(Appendix L\)](#)

Chapter 9 : Thermal / Field Elements.

2D Bar Field Elements

General



Element Group	Field
Element Subgroup	Thermal Bars
Element Description	Straight and curved
Number Of Nodes	2 or 3.
Freedom	ϕ : field value (temperature) at each node
Node Coordinates	X, Y: at each node.

Geometric Properties

$A_1 \dots A_n$ Cross-sectional area at each node.

Material Properties

Matrix	Not applicable	
Joint	Not applicable	
Composite	Not applicable	
Field	Isotropic	MATERIAL PROPERTIES FIELD ISOTROPIC (Field: Isotropic)
		MATERIAL PROPERTIES FIELD ISOTROPIC CONCRETE(Field: Isotropic)
	Orthotropic:	Not applicable
	Linear	Not applicable
	convection/radiation:	Not applicable
	Arbitrary	Not applicable
	convection/radiation:	

Loading

Prescribed Value	PDSP, TPDSP	ϕ : field variable (temperature) at nodes.
Concentrated Loads	CL	Q: field loading at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	qa: (Q/unit area) at nodes (positive defines heat input) (see FLD Face loading applied to thermal bars).
Body Forces	CBF	qv: (Q/unit volume) for element.
	BFP, BFPE	qv: (Q/unit volume) at nodes/ for element.
Velocities	Not applicable.	
Accelerations	Not applicable.	
Initial Stress/Strains	Not applicable.	
Residual Stresses	Not applicable.	
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, 0, 0, 0 (See <i>Notes</i> .)
Field Loads	ENV	Environmental temperatures . ϕ_e , hc, hr: external environmental temperature, convective and radiative heat transfer coefficients. (See <i>Notes</i>)
Temp Dependent Loads	TDET	Temperature dependent environmental temperatures . ϕ_e , hc, hr, T: external environmental temperature, convective and radiative heat transfer coefficients and temperature for element. (See <i>Notes</i>)
	RIHG	Internal heat generation rate. Q, T: coefficient/unit volume and temperature. (See <i>Notes</i>)

LUSAS Output

Solver Field variable (temperature). gx, qx: gradient and flow in local axes.

Modeller See [Results Tables \(Appendix K\)](#).

Local Axes

- ☐ [Standard line element](#)

Sign Convention

- ☐ [Standard field element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Conductivity	Default. Fine (see <i>Options</i>).	1-point (BFD2), 2-point (BFD3). 2-point (BFD2), 3-point (BFD3).
Specific Heat	Default. Fine (see <i>Options</i>).	1-point (BFD2), 2-point (BFD3). 2-point (BFD2), 3-point (BFD3).

Specific Heat Modelling

- ☐ Consistent specific heat (default).
- ☐ Lumped specific heat.

Options

- 18** Invokes fine integration rule.
- 105** Lumped specific heat.

Notes on Use

- TEMP/TMPE loading can be used to initialise the temperature field on the first step of a nonlinear field analysis. The temperature will be applied on the first pass of iteration 0 only and the load must be specified as a manual increment.
- For linear field problems only one load case is allowed if an ENVT load is to be applied.
- Load curves can be used to maintain or increment ENVT, TDET or RIHG loading as a nonlinear solution progresses.
- Automatic load incrementation under the NONLINEAR CONTROL data chapter cannot be used with TDET or RIHG loading.
- When using load curves with ENVT or TDET loading, the environmental temperatures will be factored but the heat coefficients will remain constant.

6. If radiation is to be considered the problem becomes nonlinear and NONLINEAR CONTROL must be specified.

Restrictions

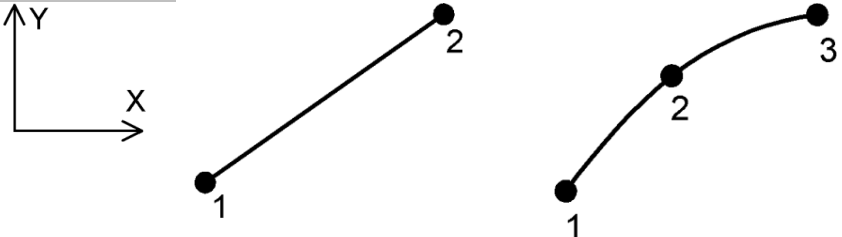
- ☐ [Ensure mid-side node centrality](#)
- ☐ Avoid excessive element curvature

Recommendations on Use

These elements may be used to analyse heat conduction along bars either individually or in conjunction with continuum field elements, e.g. supporting struts.

2D Axisymmetric Membrane Field Elements

General

Element Name	BFX2	BFX3
		
Element Group	Field	
Element Subgroup	Thermal Bars	
Element Description	Straight and curved isoparametric axisymmetric thermal bar elements in 2D which can accommodate varying cross sectional area.	
Number Of Nodes	2 or 3.	
Node Freedoms	j: field variable (temperature) at each node.	
Node Coordinates	X, Y: at each node.	

Geometric Properties

t₁... t_n Thickness at each node.

Material Properties

Matrix Not applicable.

Composite Not applicable.

Field Isotropic

MATERIAL PROPERTIES FIELD

ISOTROPIC (Field: Isotropic)

MATERIAL PROPERTIES FIELD

ISOTROPIC CONCRETE (Field: Isotropic)

Orthotropic:

Not applicable

Linear

Not applicable

convection/radiation:

Arbitrary

Not applicable

convection/radiation:

Loading

Prescribed Value	PDSP, TPDSP	ϕ : field variable (temperature) at nodes.
Concentrated Loads	CL	Q: field loading at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	qa: (Q/unit area) at nodes (positive defines heat input) (see FLD Face loading applied to thermal bars).
Body Forces	CBF	qv: (Q/unit volume) for element.
	BFP, BFPE	qv: (Q/unit volume) at nodes/ for element.
Velocities	Not applicable.	
Accelerations	Not applicable.	
Initial Stress/Strains	Not applicable.	
Residual Stresses	Not applicable.	
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, 0, 0, 0 (See <i>Notes</i> .)
Field Loads	ENVT	Environmental temperatures . ϕ_e , hc, hr: external environmental temperature, convective and radiative heat transfer coefficients. (See <i>Notes</i> .)
Temp Dependent Loads	TDET	Temperature dependent environmental temperatures . ϕ_e , hc, hr, T: external environmental temperature, convective and radiative heat transfer coefficients and temperature. (See <i>Notes</i> .)
	RIHG	Internal heat generation rate. Q, T: coefficient/unit volume and temperature for element. (See <i>Notes</i> .)

LUSAS Output

Solver Field variable (temperature). gx, qx: gradient and flow in local axes.

Modeller See [Results Tables \(Appendix K\)](#).

Local Axes

- ☐ [Standard line element](#)

Sign Convention

- ☐ Standard field element

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Conductivity	Default.	1-point (BFX2), 2-point (BFX3).
	Fine (see <i>Options</i>).	2-point (BFX2), 3-point (BFX3).
Specific Heat	Default.	1-point (BFX2), 2-point (BFX3).
	Fine (see <i>Options</i>).	2-point (BFX2), 3-point(BFX3).

Specific Heat Modelling

- ☐ Consistent specific heat (default).
- ☐ Lumped specific heat.

Options

- 18** Invokes fine integration rule.
- 47** X-axis taken as axis of symmetry.
- 105** Lumped specific heat.

Notes on Use

1. TEMP/TMPE loading can be used to initialise the temperature field on the first step of a nonlinear field analysis. The temperature will be applied on the first pass of iteration 0 only and the load must be specified as a manual increment.
2. For linear field problems only one load case is allowed if an ENVT load is to be applied.
3. Load curves can be used to maintain or increment ENVT, TDET or RIHG loading as a nonlinear solution progresses.
4. Automatic load incrementation under the NONLINEAR CONTROL data chapter cannot be used with TDET or RIHG loading.
5. When using load curves with ENVT or TDET loading, the environmental temperatures will be factored but the heat coefficients will remain constant.

6. If radiation is to be considered the problem becomes nonlinear and NONLINEAR CONTROL must be specified.

Restrictions

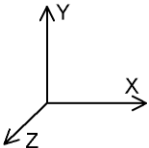
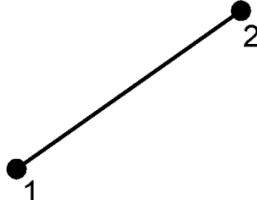
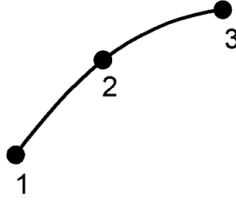
- ☐ [Ensure mid-side node centrality](#)
- ☐ Avoid excessive element curvature

Recommendations on Use

One example of the usage of these elements is the analysis of in-plane temperature flow in a thin circular plate.

3D Bar Field Elements

General

Element Name	BFS2	BFS3
		
Element Group	Field	
Element Subgroup	Thermal Bars	
Element Description	Straight and curved	
Number Of Nodes	2 or 3.	
Freedom	ϕ : field value (temperature) at each node	
Node Coordinates	X, Y, Z: at each node.	

Geometric Properties

$A_1 \dots A_n$ Cross sectional area at each node.

Material Properties

Linear	Not applicable
Matrix	Not applicable
Joint	Not applicable
Concrete	Not applicable
Elasto-Plastic	Not applicable
Creep	Not applicable
Damage	Not applicable
Viscoelastic	Not applicable
Shrinkage	Not applicable
Rubber	Not applicable.
Generic Polymer	Not applicable

Composite	Not applicable	
Field	Isotropic	MATERIAL PROPERTIES FIELD ISOTROPIC (Field: Isotropic)
		MATERIAL PROPERTIES FIELD ISOTROPIC CONCRETE (Field: Isotropic)
	Orthotropic:	Not applicable.
	Linear	Not applicable.
	convection/radiation:	
	Arbitrary	Not applicable.
	convection/radiation:	

Loading

Prescribed Value	PDSP, TPDSP	ϕ : field variable (temperature) at nodes.
Concentrated Loads	CL	Q: field loading at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	qa: (Q/unit area) at nodes (positive defines heat input) (see FLD Face loading applied to thermal bars).
Body Forces	CBF	qv: (Q/unit volume) for element.
	BFP, BFPE	qv: (Q/unit volume) at nodes/ for element.
Velocities	Not applicable.	
Accelerations	Not applicable.	
Initial Stress/Strains	Not applicable.	
Residual Stresses	Not applicable.	
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, 0, 0, 0 (See <i>Notes</i> .)
Field Loads	ENVT	Environmental temperatures . ϕ_e , hc, hr: external environmental temperature, convective and radiative heat transfer coefficients. (See <i>Notes</i> .)
Temp Dependent Loads	TDET	Temperature dependent environmental temperatures . ϕ_e , hc, hr, T: external environmental temperature, convective and radiative heat transfer coefficients and temperature. (See <i>Notes</i> .)

RIHG Internal heat generation rate. Q, T: coefficient/unit volume, and temperature for element. (See *Notes*.)

LUSAS Output

Solver Field variable (temperature). gx, qx: gradient and flow in local axes.

Modeller See [Results Tables \(Appendix K\)](#).

Local Axes

☐ [Standard line element](#)

Sign Convention

☐ [Standard field element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Conductivity	Default.	1-point (BFS2), 2-point (BFS3).
	Fine (see <i>Options</i>).	2-point (BFS2), 3-point (BFS3).
Specific Heat	Default.	1-point (BFS2), 2-point (BFS3).
	Fine (see <i>Options</i>).	2-point (BFS2), 3-point (BFS3).

Specific Heat Modelling

- ☐ Consistent specific heat (default).
- ☐ Lumped specific heat.

Options

- 18** Invokes fine integration rule.
- 105** Lumped specific heat.

Notes on Use

1. TEMP/TMPE loading can be used to initialise the temperature field on the first step of a nonlinear field analysis. The temperature will be applied on the first pass of iteration 0 only and the load must be specified as a manual increment.

2. For linear field problems only one load case is allowed if an ENVT load is to be applied.
3. Load curves can be used to maintain or increment ENVT, TDET or RIHG loading as a nonlinear solution progresses.
4. Automatic load incrementation under the NONLINEAR CONTROL data chapter cannot be used with TDET or RIHG loading.
5. When using load curves with ENVT or TDET loading, the environmental temperatures will be factored but the heat coefficients will remain constant.
6. If radiation is to be considered the problem becomes nonlinear and NONLINEAR CONTROL must be specified.

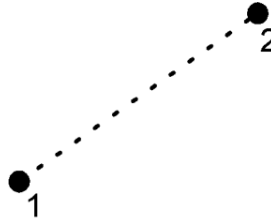
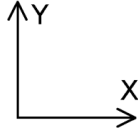
Restrictions

- ☐ [Ensure mid-side node centrality](#)
- ☐ Avoid excessive element curvature

2D Link Field Element

General

Element Name LFD2



Element Group Field

Element Thermal Links

Subgroup

Element Description Straight conductive, convective or radiative thermal link element for 2D field analysis.

Number Of Nodes 2.

Freedom ϕ : field value (temperature) at each node.

Node Coordinates X, Y at each node.

Geometric Properties

$A_1 \dots A_n$ Cross sectional area at each node.

Material Properties

Linear Not applicable

Matrix Not applicable

Joint Not applicable

Concrete Not applicable

Elasto-Plastic Not applicable

Creep Not applicable

Damage Not applicable

Viscoelastic Not applicable

Shrinkage Not applicable

Rubber Not applicable

Generic Not applicable

Polymer

Composite Not applicable

Field Isotropic:	Not applicable.
Orthotropic:	Not applicable.
Linear	MATERIAL PROPERTIES FIELD
convection/radiation:	LINK 18 (Field: Linear Link)
Arbitrary	MATERIAL PROPERTIES FIELD
convection/radiation:	LINK 19 (Field: Nonlinear Link)

Loading

Prescribed Value	PDSP, TPDSP	ϕ : field variable (temperature) at nodes.
Concentrated Loads	Not applicable.	
Element Loads	Not applicable.	
Distributed Loads	Not applicable.	
Body Forces	Not applicable.	
Velocities	Not applicable.	
Accelerations	Not applicable.	
Initial Stress/Strains	Not applicable.	
Residual Stresses	Not applicable.	
Temperatures	Not applicable.	
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver Field variable (temperature). qx: flow at nodes in local directions.

Modeller See [Results Tables \(Appendix K\)](#).

Local Axes

- ☐ [Standard line element](#)

Sign Convention

- ☐ [Standard field element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Conduction, Convection, Radiation	Default.	1-point (at element centroid).
	Fine.	As default.
Specific Heat	Default.	Not applicable.
	Fine.	Not applicable.

Specific Heat Modelling

Not applicable.

Options

Not applicable.

Notes on Use

No notes at present.

Restrictions

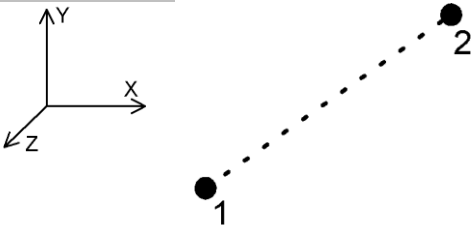
Not applicable.

Recommendations on Use

An example of the usage of these elements is the analysis of heat conduction at contacting interfaces.

3D Link Field Element

General

Element Name	LFS2
	
Element Group	Field
Element Subgroup	Thermal Links
Element Description	Straight conductive, convective or radiative thermal link element for 3D field analysis.
Number Of Nodes	2.
End Releases	
Freedom	ϕ : field value (temperature) at each node.
Node Coordinates	X, Y, Z at each node.

Geometric Properties

$A_1 \dots A_n$ Cross sectional area at each node.

Material Properties

Linear	Not applicable.	
Matrix	Not applicable.	
Joint	Not applicable.	
Concrete	Not applicable.	
Elasto-Plastic	Not applicable.	
Rubber	Not applicable.	
Generic Polymer	Not applicable.	
Composite	Not applicable.	
Field	Isotropic:	Not applicable.
	Orthotropic:	Not applicable.

	Linear convection/radiation:	MATERIAL PROPERTIES FIELD LINK 18 (Field: Linear Link)
	Arbitrary convection/radiation:	MATERIAL PROPERTIES FIELD LINK 19 (Field: Nonlinear Link)
Stress	Not applicable.	
Potential		
Creep	Not applicable.	
Damage	Not applicable.	
Viscoelastic	Not applicable.	
Shrinkage	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	φ : field variable (temperature) at nodes.
Concentrated Loads	Not applicable.	
Element Loads	Not applicable.	
Distributed Loads	Not applicable.	
Body Forces	Not applicable.	
Velocities	Not applicable.	
Accelerations	Not applicable.	
Initial Stress/Strains	Not applicable.	
Residual Stresses	Not applicable.	
Temperatures	Not applicable.	
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Field variable (temperature). qx: flow at nodes in local directions.
Modeller	See Results Tables (Appendix K) .

Local Axes

- ☐ [Standard line element](#)

Sign Convention

- ☐ [Standard field element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Conduction, Convection, Radiation	Default.	1- point (at element centroid).
	Fine.	As default.
Specific Heat	Default.	Not applicable.
	Fine.	Not applicable.

Specific Heat Modelling

Not applicable.

Options

Not applicable.

Notes on Use

No notes at present.

Restrictions

Not applicable.

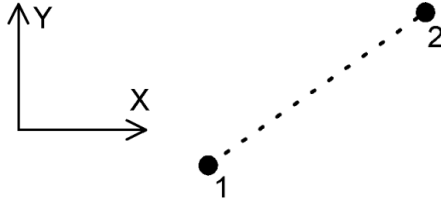
Recommendations on Use

An example of the usage of these elements is the analysis of heat conduction at contacting interfaces.

2D Axisymmetric Link Field Element

General

Element Name LFX2



Element Group	Field
Element Subgroup	Thermal Links
Element Description	Straight conductive, convective or radiative thermal link element for 2D axisymmetric field analysis.
Number Of Nodes	2.
End Releases	
Freedom	ϕ : field value (temperature) at each node.
Node Coordinates	X, Y at each node.

Geometric Properties

$t_1 \dots t_n$ Thickness at each node.

Material Properties

Linear	Not applicable.	
Matrix	Not applicable.	
Joint	Not applicable.	
Concrete	Not applicable.	
Elasto-Plastic	Not applicable.	
Rubber	Not applicable.	
Generic Polymer	Not applicable.	
Composite	Not applicable.	
Field	Isotropic:	Not applicable.
	Orthotropic:	Not applicable.
	Linear	MATERIAL PROPERTIES FIELD LINK

convection/radiation:	18 (Field: Linear Link)
Arbitrary	MATERIAL PROPERTIES FIELD LINK
convection/radiation:	19 (Field: Nonlinear Link)

Loading

Prescribed Value	PDSP, TPDSP	φ : field variable (temperature) at nodes.
Concentrated Loads	Not applicable.	
Element Loads	Not applicable.	
Distributed Loads	Not applicable.	
Body Forces	Not applicable.	
Velocities	Not applicable.	
Accelerations	Not applicable.	
Initial Stress/Strains	Not applicable.	
Residual Stresses	Not applicable.	
Temperatures	Not applicable.	
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Field variable (temperature). qx: flow at nodes in local directions.
Modeller	See Results Tables (Appendix K) .

Local Axes

- ☐ [Standard line element](#)

Sign Convention

- ☐ [Standard field element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Conduction, Convection, Radiation	Default.	1- point (at element centroid).
	Fine.	As default.

Specific Heat	Default.	Not applicable.
	Fine.	Not applicable.

Specific Heat Modelling

Not applicable.

Options

- 47 X-axis taken as axis of symmetry.

Notes on Use

No notes at present.

Restrictions

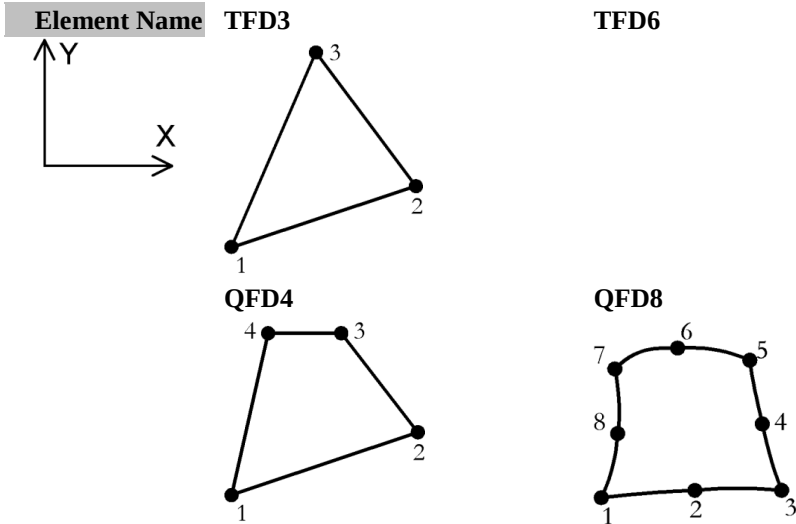
Not applicable.

Recommendations on Use

An example of the usage of these elements is the analysis of heat conduction at contacting interfaces.

2D Plane Field Elements

General



Element Group	Field
Element Subgroup	Plane Field
Element Description	A family of plane field elements in 2D with higher order elements capable of modelling curved boundaries. The elements are applicable to both steady state and transient field problems. The elements are numerically integrated.
Number Of Nodes	3, 4, 6 or 8 numbered anticlockwise.
Freeds	ϕ : field value (temperature) at each node.
Node Coordinates	X, Y: at each node.

Geometric Properties

$t_1 \dots t_n$ Thickness at each node.

Material Properties

Linear	Not applicable
Matrix	Not applicable
Joint	Not applicable

Concrete	Not applicable	
Elasto-Plastic	Not applicable	
Creep	Not applicable	
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage	Not applicable	
Rubber	Not applicable.	
Generic Polymer	Not applicable	
Composite	Not applicable.	
Field	Isotropic:	MATERIAL PROPERTIES FIELD ISOTROPIC CONCRETE (Field: Isotropic)
		MATERIAL PROPERTIES FIELD ISOTROPIC (Field: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES FIELD ORTHOTROPIC (Field: Orthotropic)
		MATERIAL PROPERTIES FIELD ORTHOTROPIC CONCRETE (Field: Orthotropic)
	Linear	Not applicable.
	convection/radiation:	
	Arbitrary	Not applicable.
	convection/radiation:	

Loading

Prescribed Value	PDSP, TPDSP	ϕ : field variable (temperature) at nodes.
Concentrated Loads	CL	Q: field loading at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	qa: (Q/unit area) at nodes (see FLD Face loading applied to thermal bars).
Body Forces	CBF	qv: (Q/unit volume) for element.
	BFP, BFPE	qv: (Q/unit volume) at nodes/ for element.
Velocities	Not applicable.	
Accelerations	Not applicable.	

Initial	Not	
Stress/Strains	applicable.	
Residual	Not	
Stresses	applicable.	
Temperatures	Not	
	applicable.	
Field Loads	ENVT	Environmental temperatures . ϕ_e , hc, hr: external environmental temperature, convective and radiative heat transfer coefficients. (See <i>Notes</i> .)
Temp	TDET	Temperature dependent environmental
Dependent		temperatures . ϕ_e , hc, hr, T: external environmental temperature, convective and radiative heat transfer coefficients and temperature. (See <i>Notes</i> .)
Loads		
	RIHG	Internal heat generation rate. Q, T: coefficient/unit volume and temperature for element. (See <i>Notes</i> .)

LUSAS Output

Solver	Field variable (temperature). gx, gy, qx, qy: gradients and flows in global directions.
Modeller	See Results Tables (Appendix K) .

Local Axes

- ☐ [Standard area element](#)

Sign Convention

- ☐ [Standard field element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Conductivity	Default.	1-point (TFD3), 3-point (TFD6), 2x2 (QFD4, QFD8).
	Fine.	As default.
Specific Heat	Default.	1-point (TFD3), 3-point (TFD6), 2x2 (QFD4, QFD8).
	Fine.	Not applicable.

Specific Heat Modelling

- ☐ Consistent specific heat (default).
- ☐ Lumped specific heat.

Options

- 18** Invokes fine integration rule for elements.
- 105** Lumped specific heat.

Notes on Use

1. The element formulations are based on the standard [isoparametric](#) approach. The variation of field variable (temperature) within an element is linear for low order (corner node only) elements and quadratic for high order (mid-side node) elements.
2. All elements pass the [patch test](#) for convergence.
3. For linear field problems only one load case is allowed if an ENVT load is to be applied.
4. Load curves can be used to maintain or increment ENVT, TDET or RIHG loading as a nonlinear solution progresses.
5. Automatic load incrementation under the NONLINEAR CONTROL data chapter cannot be used with TDET or RIHG loading.
6. When using load curves with ENVT or TDET loading, the environmental temperatures will be factored but the heat coefficients will remain constant.
7. If radiation is to be considered the problem becomes nonlinear and NONLINEAR CONTROL must be specified.

Restrictions

- ☐ [Ensure mid-side node centrality](#)
- ☐ [Avoid excessive element curvature](#)
- ☐ Avoid excessive aspect ratio

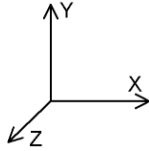
Recommendations on Use

The plane field elements may be utilised for analysing continuum field problems whose behaviour is essentially two dimensional, e.g. thermal analysis of a long tunnel . The elements are formulated using the 2D quasi-harmonic equation. See Theory Manuals for details.

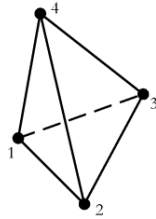
3D Solid Field Elements

General

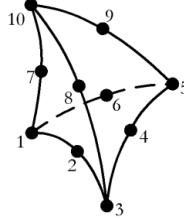
Element Name



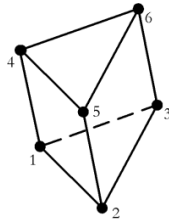
TF4



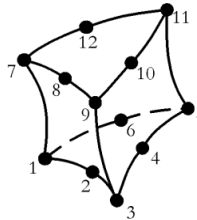
TF10



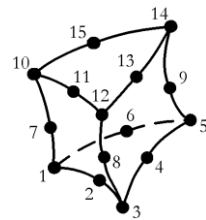
PF6



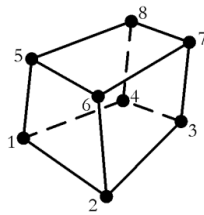
PF12



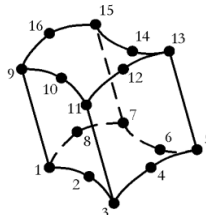
PF15



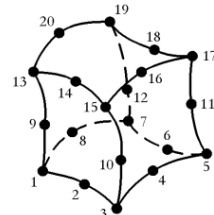
HF8



HF16



HF20



Element Group

Field

Element

Solid Field

Subgroup

Element

Description

A family of solid field elements in 3D with higher order elements capable of modelling curved boundaries. The elements are applicable to both steady state and transient field problems. The elements are numerically integrated.

Number Of

Nodes

4 and 10 (tetrahedra). 6, 12 and 15 (pentahedra). 8, 16 and 20 (hexahedra). The elements are numbered according to a right-hand screw rule in the local z-direction.

Freedom

ϕ : field variable at each node.

Node

X, Y, Z: at each node.

Coordinates

Geometric Properties

Not applicable.

Material Properties

Linear	Not applicable	
Matrix	Not applicable	
Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic	Not applicable	
Creep	Not applicable	
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage	Not applicable	
Rubber	Not applicable	
Generic Polymer	Not applicable	
Composite	Not applicable	
Field	Isotropic:	MATERIAL PROPERTIES FIELD ISOTROPIC CONCRETE (Field: Isotropic)
		MATERIAL PROPERTIES FIELD ISOTROPIC (Field: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES FIELD ORTHOTROPIC SOLID (Field: Orthotropic Solid)
		MATERIAL PROPERTIES FIELD ORTHOTROPIC SOLID CONCRETE (Field: Orthotropic Solid)
	Linear convection/radiation:	Not applicable.
	Arbitrary convection/radiation:	Not applicable.

Loading

Prescribed Value	PDSP, TPDSP	ϕ : field variable (temperature) at nodes.
Concentrated Loads	CL	Q: field loading at nodes.
Element Loads	Not applicable.	

Distributed Loads	UDL	Not applicable.
	FLD	qa: (Q/unit area) at nodes (see FLD Face loading applied to thermal bars).
Body Forces	CBF	qv: (Q/unit volume) for element.
	BFP, BFPE	qv: (Q/unit volume) at nodes/ for element.
Velocities	Not applicable.	
Accelerations	Not applicable.	
Initial Stress/Strains	Not applicable.	
Residual Stresses	Not applicable.	
Temperatures	Not applicable.	
Field Loads	ENVT	Environmental temperatures . ϕ_e , hc, hr: external environmental temperature, convective and radiative heat transfer coefficients. (See <i>Notes</i> .)
	Temp Dependent Loads	Temperature dependent environmental temperatures . ϕ_e , hc, hr, T: external environmental temperature, convective and radiative heat transfer coefficients and temperature. (See <i>Notes</i> .)
	RIHG	Internal heat generation rate. Q, T: coefficient/unit volume and temperature for element. (See <i>Notes</i> .)

LUSAS Output

Solver	Field variable (temperature). gx, gy, gz, qx, qy, qz: gradients and flows in global directions.
Modeller	See Results Tables (Appendix K) .

Local Axes

Not applicable (global axes are the reference).

Sign Convention

- ☐ [Standard field element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Conductivity	Default.	1-point (TF4), 4-point (TF10), 3x2 (PF6, PF12, PF15), 2x2x2 (HF8, HF16, HF20)
	Fine (see <i>Options</i>).	5-point (TF10) 3x3x2 (HF16), 3x3x3 (HF20)
	Coarse (see <i>Options</i>).	1-point (HF20), 14-point (HF20)
Specific Heat	Default.	1-point (TF4), 4-point (TF10), 3x2 (PF6, PF12, PF15), 2x2x2 (HF8, HF16, HF20)
	Fine (see <i>Options</i>).	5-point (TF10) 3x3x2 (HF16), 3x3x3 (HF20)
	Coarse (see <i>Options</i>).	13-point (HF20), 14-point (HF20)

Specific Heat Modelling

- ☐ Consistent specific heat (default).
- ☐ Lumped specific heat.

Options

- 18** Invokes fine integration rule for elements.
- 105** Lumped specific heat.
- 155** Use 14-point integration rule for HF20.
- 156** Use 13-point integration rule for HF20.
- 398** For HF20 and HF16 with fine integration use all integration points for stress extrapolation.

Notes on Use

1. The element formulations are based on the standard isoparametric approach. The variation of potential within an element may be regarded as constant for low order (corner node only) elements and linear for high order (mid-side node) elements.

2. For linear field problems only one load case is allowed if an ENVT load is to be applied.
3. Load curves can be used to maintain or increment ENVT, TDET or RIHG loading as a nonlinear solution progresses.
4. Automatic load incrementation under the NONLINEAR CONTROL data chapter cannot be used with TDET or RIHG loading.
5. When using load curves with ENVT or TDET loading, the environmental temperatures will be factored but the heat coefficients will remain constant.
6. If radiation is to be considered the problem becomes nonlinear and NONLINEAR CONTROL must be specified.

Restrictions

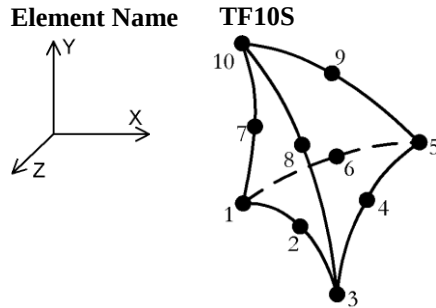
- ☐ [Ensure mid-side node centrality](#)
- ☐ [Avoid excessive element curvature](#)
- ☐ Avoid excessive aspect ratio

Recommendations on Use

The solid field elements may be used to analyse continuum field problems where the response is fully 3D (i.e. it cannot be approximated using the plane or axisymmetric elements), e.g. temperature distribution in a pipe intersection.

3D Solid Composite Field Element (Tetrahedral)

General



Element Group	Field
Element Subgroup	Solid Field
Element Description	3D solid field element capable of modelling curved boundaries. The element is applicable to both steady state and transient field problems. The element is numerically integrated, can be arbitrarily oriented with respect to the laminate, and allows for the fully automatic mesh generation of laminate geometric models imported from CAD packages.
Number Of Nodes	10. The element is numbered according to a right-hand screw rule in the local z-direction.
Freedoms	ϕ : field variable at each node.
Node Coordinates	X, Y, Z: at each node.

Geometric Properties

See [Composites](#) in the *Modeller Reference Manual*

Material Properties

Linear	Not applicable
Matrix	Not applicable
Joint	Not applicable
Concrete	Not applicable
Elasto-Plastic	Not applicable
Creep	Not applicable
Damage	Not applicable

Viscoelastic	Not applicable	
Shrinkage	Not applicable	
Rubber	Not applicable	
Generic	Not applicable	
Polymer		
Composite		COMPOSITE MATERIAL
Field	Isotropic:	MATERIAL PROPERTIES FIELD ISOTROPIC (Field: Isotropic) MATERIAL PROPERTIES FIELD ISOTROPIC CONCRETE (Field: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES FIELD ORTHOTROPIC SOLID (Field: Orthotropic Solid) MATERIAL PROPERTIES FIELD ORTHOTROPIC SOLID CONCRETE (Field: Orthotropic Solid)
	Linear convection/radiation:	Not applicable
	Arbitrary convection/radiation:	Not applicable

Loading

Prescribed Value	PDSP, TPDSP	ϕ : field variable (temperature) at nodes.
Concentrated Loads	CL	Q: field loading at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	qa: (Q/unit area) at nodes
Body Forces	CBF	qv: (Q/unit volume) for element.
	BFP, BFPE	qv: (Q/unit volume) at nodes/ for element.
Velocities	Not applicable.	
Accelerations	Not applicable.	
Initial Stress/Strains	Not applicable.	
Residual Stresses	Not applicable.	
Temperatures	Not	

		applicable.
Field Loads	ENV T	Environmental temperatures ϕ_e , h_c , h_r : external environmental temperature, convective and radiative heat transfer coefficients. (See <i>Notes</i> .)
Temp Dependent Loads	TDET	Temperature dependent environmental temperatures. ϕ_e , h_c , h_r , T : external environmental temperature, convective and radiative heat transfer coefficients and temperature. (See <i>Notes</i> .)
	RIHG	Internal heat generation rate. Q , T : coefficient/unit volume and temperature for element. (See <i>Notes</i> .)

LUSAS Output

Solver Field variable (temperature). g_x , g_y , g_z , q_x , q_y , q_z : gradients and flows. Gauss point values are in local directions. Nodal values are in global directions.

Modeller See [Results tables \(Appendix K\)](#)

Local Axes

The local axes for each layer are defined by the LAMINAR DIRECTIONS specified for its bottom surface. The three node set in LAMINAR DIRECTIONS define the local Cartesian set origin, the x-axis and the positive quadrant of the xy-plane respectively. The local z-axis forms an orthonormal coordinate system with x and y.

Sign Convention

☐ [Standard field elements](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Conductivity	Default.	1-point for a tetrahedral subdivision (see <i>Notes</i>), 3-point for a pentahedral/pyramid subdivision, 2x2 for a hexahedral/wrick subdivision
	Fine (see <i>Options</i>).	1-point for a tetrahedral subdivision (see <i>Notes</i>), 3x2 for a pentahedral/pyramid subdivision, 2x2 x2 for a hexahedral/wrick subdivision
Specific Heat	Default.	5-point for the whole element or (see <i>Options</i>) 1-

point for a tetrahedral subdivision, 3x2 for a pentahedral/pyramid subdivision, 2x2 x2 for a hexahedral/wrick subdivision

Fine (see 11-point or (see Options) 14 -point for the whole *Options*). element

Specific Heat Modelling

- Consistent specific heat (default).
- Lumped specific heat.

Options

- 18** Invokes fine integration rule for elements.
- 91** Formulate element specific heat with fine integration
- 105** Lumped specific heat.
- 266** Layer by layer computation of specific heat matrix.
- 394** Lamina directions supported
- 395** Use 14-point fine integration rule for specific heat matrix of TH10 family (used together with 91)

Notes on Use

1. The element formulations are based on the standard isoparametric approach. The variation of field gradients within an element may be regarded as linear.
2. The LAMINAR DIRECTIONS and COMPOSITE MATERIAL data chapters must be used with this element in conjunction with the COMPOSITE ASSIGNMENTS data chapter.
3. If the whole tetrahedral element is embedded in a single lamina, a 4-point integration rule will be used for this tetrahedral subdivision; otherwise a 1-point rule will be used.
4. The specific heat matrix can be computed using a layer by layer integration (OPTION 266), however this should only be used when the thermal properties of the layers vary considerably because the computation time can be greatly increased when this OPTION is specified.
5. Numerical integration through the thickness is performed. The integration points are located in the subdivisions of each layer. Each subdivision forms the shape of a regular 3D solid field element and the integration points are located accordingly within the subdivision as described above.

6. For linear field problems only one load case is allowed if an ENVT load is to be applied.
7. Load curves can be used to maintain or increment ENVT, TDET or RIHG loading as a nonlinear solution progresses.
8. Automatic load incrementation under the NONLINEAR CONTROL data chapter cannot be used with TDET or RIHG loading.
9. When using load curves with ENVT or TDET loading, the environmental temperatures will be factored but the heat coefficients will remain constant.
10. If radiation is to be considered the problem becomes nonlinear and NONLINEAR CONTROL must be specified.
11. Layer 1 is always the bottom layer.

Restrictions

- ☐ [Ensure mid-side node centrality](#)
- ☐ [Avoid excessive element curvature](#)
- ☐ [Avoid excessive aspect ratio](#)

Recommendations on Use

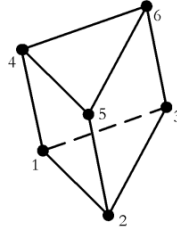
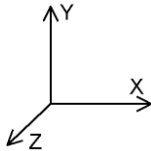
- 3D solid composite field elements should be used for modelling thick composite structures comprising laminae of differing material properties where the computational cost of modelling each lamina with an individual solid element would be prohibitive. This field element can be used to analyse continuum field problems where the response is fully 3D.
- As these elements can be arbitrarily oriented with respect to the laminate, they are particularly aimed at the use of fully automatic mesh generation of laminate geometric models imported from CAD packages.

3D Solid Composite Field Elements (Pentahedral and Hexahedral)

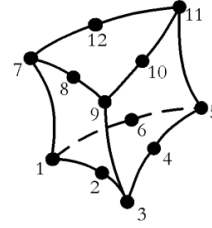
General

Element Name

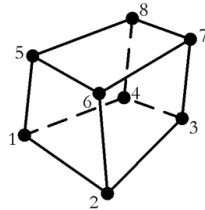
PF6C



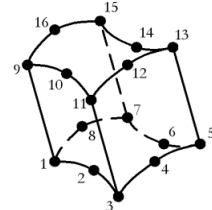
PF12C



HF8C



HF16C



Element Group

Field

Element

Solid Field

Subgroup

Element

Description

3D solid field elements capable of modelling curved boundaries. The elements are applicable to both steady state and transient field problems. The elements are numerically integrated. The composite layers are parallel to the top and bottom faces and the bottom surface of the first layer coincides with the bottom surface of the element. The top and bottom faces of the element are as shown, e.g. nodes 1, 2, 3, 4 define the bottom face of HF8C

Number Of
Nodes

6 or 12 (pentahedra), 8 or 16 (hexahedra). The elements are numbered according to a right-hand screw rule in the local z-direction.

Freedom

ϕ : field variable at each node.

Node

X, Y, Z: at each node.

Coordinates

Geometric Properties

See [Composites](#) in the *Modeller Reference Manual*

Material Properties

Linear	Not applicable	
Matrix	Not applicable	
Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic	Not applicable	
Creep	Not applicable	
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage	Not applicable	
Rubber	Not applicable	
Generic Polymer	Not applicable	
Composite		COMPOSITE MATERIAL
Field	Isotropic:	MATERIAL PROPERTIES FIELD ISOTROPIC (Field: Isotropic) MATERIAL PROPERTIES FIELD ISOTROPIC CONCRETE (Field: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES FIELD ORTHOTROPIC SOLID (Field: Orthotropic Solid) MATERIAL PROPERTIES FIELD ORTHOTROPIC SOLID CONCRETE (Field: Orthotropic Solid)
	Linear convection/radiation:	Not applicable
	Arbitrary convection/radiation:	Not applicable

Loading

Prescribed Value	PDSP, TPDSP	φ : field variable (temperature) at nodes.
Concentrated Loads	CL	Q: field loading at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	qa: (Q/unit area) at nodes
Body Forces	CBF	qv: (Q/unit volume) for element.

	BFP, BFPE	qv: (Q/unit volume) at nodes/ for element.
Velocities	Not applicable.	
Accelerations	Not applicable.	
Initial Stress/Strains	Not applicable.	
Residual Stresses	Not applicable.	
Temperatures	Not applicable.	
Field Loads	ENVT	Environmental temperatures ϕ_e , h_c , h_r : external environmental temperature, convective and radiative heat transfer coefficients. (See <i>Notes</i> .)
Temp Dependent Loads	TDET	Temperature dependent environmental temperatures. ϕ_e , h_c , h_r , T : external environmental temperature, convective and radiative heat transfer coefficients and temperature. (See <i>Notes</i> .)
	RIHG	Internal heat generation rate. Q , T : coefficient/unit volume and temperature for element. (See <i>Notes</i> .)

LUSAS Output

Solver Field variable (temperature). g_x , g_y , g_z , q_x , q_y , q_z : gradients and flows. Gauss point values are in local directions. Nodal values are in global directions.

Modeller See [Results tables \(Appendix K\)](#)

Local Axes

The local axes for each layer are defined using the convention for [standard area elements](#). Local axes are computed at the top and bottom quadratic surfaces (at the Gauss points) and average values are interpolated for the mid-surface. Every layer uses the same averaged values.

Sign Convention

- [Standard field elements](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Conductivity	Default.	1-point for each layer (PF6C), 3-point for each layer (PF12C), 2x2 for each layer (HF8C, HF16C)
	Fine (see <i>Options</i>).	3-point for each layer (PF6C), 3x3 for each layer (HF16C)
Specific Heat	Default.	3x2 for the whole element (PF6C, PF12C) or (see <i>Options</i>) 1-point for each layer (PF6C), 3-point for each layer (PF12C), 2x2x2 for the whole element or 2x2 for each layer (HF8C, HF16C)
	Fine (see <i>Options</i>).	3x2 for the whole element or 3-point for each layer (PF6C), 3x3x2 for the whole element or 3x3 for each layer (HF16C)

Specific Heat Modelling

- Consistent specific heat (default).
- Lumped specific heat.

Options

- 18** Invokes fine integration rule for elements.
- 105** Lumped specific heat.
- 266** Layer by layer computation of specific heat matrix.

Notes on Use

1. The element formulations are based on the standard isoparametric approach.
2. For linear field problems only one load case is allowed if an ENVT load is to be applied.
3. The COMPOSITE GEOMETRY and COMPOSITE MATERIAL data chapters must be used with this element in conjunction with the COMPOSITE ASSIGNMENTS data chapter.
4. Load curves can be used to maintain or increment ENVT, TDET or RIHG loading as a nonlinear solution progresses.

5. Automatic load incrementation under the NONLINEAR CONTROL data chapter cannot be used with TDET or RIHG loading.
6. When using load curves with ENVT or TDET loading, the environmental temperatures will be factored but the heat coefficients will remain constant.
7. If radiation is to be considered the problem becomes nonlinear and NONLINEAR CONTROL must be specified.
8. The through thickness integration is performed assuming a linear variation of the field gradient-variable matrix for each layer.
9. Layer 1 is always the bottom layer.
10. The simplifying assumptions which allow the uncoupling of in-plane and through thickness co-ordinates leads to the restriction that any individual layer should be of a constant thickness. This restriction should be considered when the finite element mesh is created and adhered to as closely as possible. In addition, out of plane lamina curvatures should also be minimised although in-plane curvature (in the x-y plane) is not restricted.

Restrictions

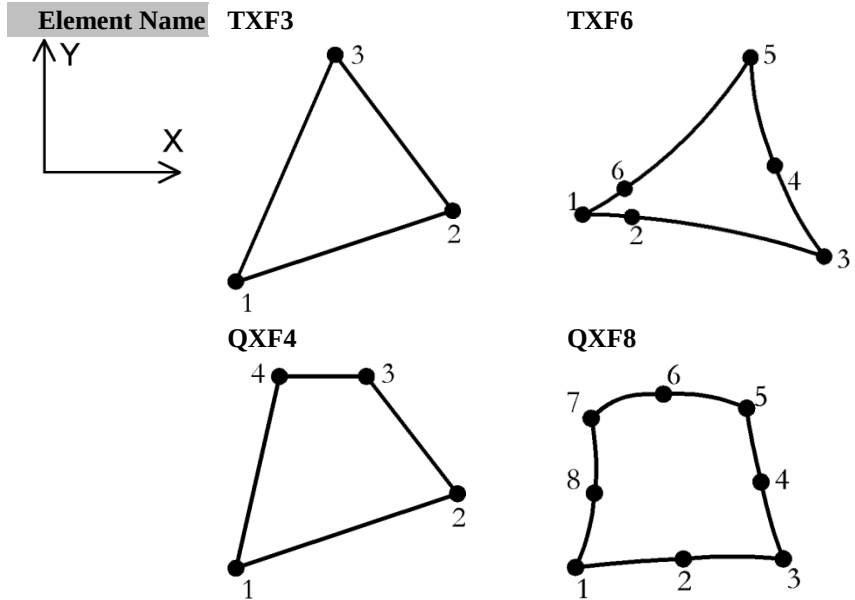
- ☐ [Ensure mid-side node centrality](#)
- ☐ [Avoid excessive element curvature](#)
- ☐ [Avoid excessive aspect ratio](#)
- ☐ Constant layer thickness for each individual layer

Recommendations on Use

The 3D solid composite field elements should be used for modelling thick composite structures comprising laminae of differing material properties where the computational cost of modelling each lamina with an individual solid element would be prohibitive. These field elements can be used to analyse continuum field problems where the response is fully 3D.

2D Axisymmetric Field Elements

General



Element Group	Field
Element Subgroup	Plane Field
Element Description	A family of axisymmetric field elements in 2D with higher order elements capable of modelling curved boundaries. The elements are applicable to both steady state and transient field problems. The formulations apply over a unit radian segment of the structure and the loading and boundary conditions are axisymmetric. The elements are numerically integrated. Axisymmetry is taken about the Y-axis by default.
Number Of Nodes	3, 4, 6, or 8 numbered anticlockwise.
Freedom	ϕ : field variable at each node.
Node Coordinates	X, Y: at each node

Geometric Properties

Not applicable (a unit radian segment is assumed).

Material Properties

Linear	Not applicable.	
Matrix	Not applicable.	
Joint	Not applicable.	
Concrete	Not applicable.	
Elasto-Plastic	Not applicable.	
Rubber	Not applicable.	
Generic Polymer	Not applicable.	
Composite	Not applicable.	
Field	Isotropic:	MATERIAL PROPERTIES FIELD ISOTROPIC (Field: Isotropic)
		MATERIAL PROPERTIES FIELD ISOTROPIC CONCRETE (Field: Isotropic)
	Orthotropic:	MATERIAL PROPERTIES FIELD ORTHOTROPIC (Field: Orthotropic)
		MATERIAL PROPERTIES FIELD ORTHOTROPIC CONCRETE (Field: Orthotropic)
	Linear convection/radiation:	Not applicable.
	Arbitrary convection/radiation:	Not applicable.

Loading

Prescribed Value	PDSP, TPDSP	ϕ : field variable (temperature) at nodes.
Concentrated Loads	CL	Q: field loading at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Not applicable.
	FLD	qa: (Q/unit area) at nodes (see FLD Face loading applied to thermal bars).
Body Forces	CBF	qv: (Q/unit volume) for element.
	BFP, BFPE	qv: (Q/unit volume) at nodes/ for element.
Velocities	Not applicable.	

Accelerations	Not applicable.	
Initial Velocities	Not applicable.	
Initial Stress/Strains	Not applicable.	
Residual Stresses	Not applicable.	
Temperatures	Not applicable.	
Field Loads	ENVT	Environmental temperatures . ϕ_e , hc, hr: external environmental temperature, convective and radiative heat transfer coefficients. (See <i>Notes</i> .)
Temp Dependent Loads	TDET	Temperature dependent environmental temperatures . ϕ_e , hc, hr, T: external environmental temperature, convective and radiative heat transfer coefficients and temperature. (See <i>Notes</i> .)
	RIHG	Internal heat generation rate. Q, T: coefficient/unit volume and temperature for element. (See <i>Notes</i> .)

LUSAS Output

Solver	Field variable (temperature). gx, gy, gz, qx, qy, qz: gradients and flows in global directions.
Modeller	See Results Tables (Appendix K) .

Local Axes

Not applicable.

Sign Convention

☐ [Standard field element](#)

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Conductivity	Default.	1-point (TXF3), 3-point (TXF6), 2x2 (QXF4, QXF8)

	Fine (see <i>Options</i>).	3x3 (QXF8)
Specific Heat	Default.	1-point (TXF3), 3-point (TXF6), 2x2 (QXF4, QXF8)
	Fine.	As default.

Specific Heat Modelling

- ☐ Consistent specific heat (default)
- ☐ Lumped specific heat.

Options

- 18** Invokes fine integration rule for elements.
- 47** X-axis taken as axis of symmetry.
- 105** Lumped specific heat.

Notes on Use

1. The element formulations are based on the standard [isoparametric](#) approach. The variation of field variable (temperature) within an element is linear low order (corner node only) elements and quadratic high order (mid-side node) elements.
2. All elements pass the [patch test](#) for convergence.
3. For linear field problems only one load case is allowed if an ENVT load is to be applied.
4. Load curves can be used to maintain or increment ENVT, TDET or RIHG loading as a nonlinear solution progresses.
5. Automatic load incrementation under the NONLINEAR CONTROL data chapter cannot be used with TDET or RIHG loading.
6. When using load curves with ENVT or TDET loading, the environmental temperatures will be factored but the heat coefficients will remain constant.
7. If radiation is to be considered the problem becomes nonlinear and NONLINEAR CONTROL must be specified.

Restrictions

- ☐ [Ensure mid-side node centrality](#)
- ☐ [Avoid excessive element curvature](#)
- ☐ Avoid excessive aspect ratio

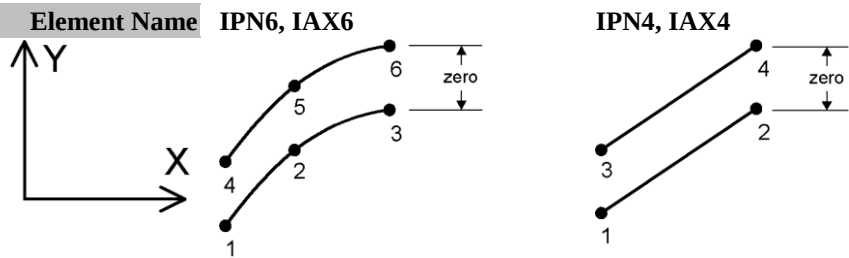
Recommendations on Use

The axisymmetric field elements are suitable for analysing solid field problems which exhibit geometric and loading symmetry about a given axis, e.g. temperature distribution in a pipe or radial groundwater flow into a well.

Chapter 10 : Interface Elements.

2D Interface Element

General



Element Group	Interface
Element Subgroup	2D Interface
Element Description	A family of 2D interface elements used for modelling delamination for plane and axisymmetric and crack propagation.
Number Of Nodes	4,6
Freedoms	U, V: at each node.
Node Coordinates	X, Y: at each node.

Geometric Properties

Not applicable (a zero thickness is assumed).

Material Properties

Linear	Not applicable
Matrix	Not applicable
Joint	Not applicable
Concrete	Not applicable
Elasto-Plastic	Not applicable
Creep	Not applicable
Damage	Not applicable
Viscoelastic	Not applicable
Shrinkage	Not applicable
Interface	Interface
Rubber	Not applicable
Generic Polymer	Not applicable
Composite	Not applicable

MATERIAL PROPERTIES NONLINEAR 25

Loading

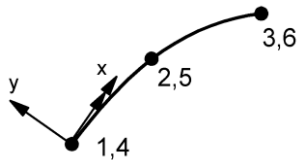
Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V: at each node.
Concentrated Loads	CL	Concentrated loads. Px, Py: at each node.
Element Loads	Not applicable.	
Distributed Loads	Not applicable.	
Body Forces	Not applicable.	
Velocities	VELO	Velocities. Vx, Vy: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay: at nodes.
Initial Stress/Strains	Not applicable.	
Residual Stresses	Not applicable.	
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

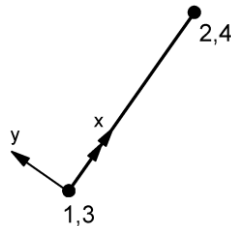
Solver	Stress (default): shear and direct tractions. Strain: shear and direct relative displacements
Modeller	See Results Tables (Appendix K) .

Local Axes

Element Name IPN6,IAX6
Evaluated at
each node.



IPN4,IAX4



Sign Convention

A positive traction occurs if the local relative displacement (with respect to the first line of the element) is a positive value, i.e. for the quadratic elements at nodes $3 > 6$ the local relative displacement, Ez , would be positive if $(Dz3 - Dz6) > 0$, where Dzi is the local displacement at node i .

Formulation

Geometric Nonlinearity

Total Lagrangian	Not applicable.
Updated Lagrangian	Not applicable.
Eulerian	Not applicable.
Co-rotational	Applicable to IPN4 and IAX4 elements.

Integration Schemes

Stiffness	Default.	3 (Newton-Cotes) (IPN6,IAX6), 2 (Newton Cotes) (IPN4,IAX4)
	Fine.	As default

Mass Modelling

Not applicable.

Options

- 62 Continue solution if more than one negative pivot occurs
- 229 Co-rotational geometric non-linearity.
- 252 Suppress pivot warning messages.
- 261 Select the root with the lowest residual norm with arc-length.

Notes on Use

1. When defining the transient analysis control the arc-length procedure should be adopted with the option to select the root with the lowest residual norm [option 261].
2. It is recommended that fine integration [option 18] is selected for the parent elements.
3. The nonlinear convergence criteria should be selected to converge on the residual norm.
4. Option 62, Continue solution if more than one negative pivot occurs, should be selected to continue if more than one negative pivot is encountered and option 252 should be used to suppress pivot warning messages from the solution process.
5. The non-symmetric solver is selected automatically when mixed mode delamination is specified.
6. Although the solution is largely independent of the mesh discretisation, to avoid convergence difficulties it is recommended that at least 2 elements are placed in the process zone.

Restrictions

None.

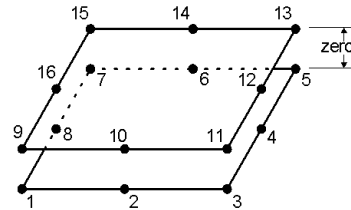
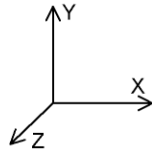
Recommendations on Use

These elements should be used at places of potential delamination between 2D plane and axisymmetric continuum elements.

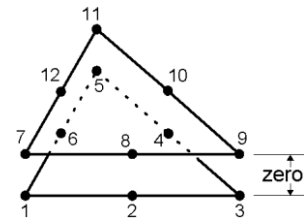
3D Interface Element

General

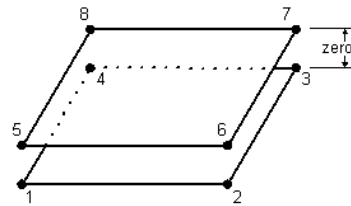
Element Name IS16



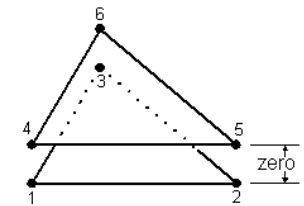
IS12



IS8



IS6



Element Group	Interface
Element Subgroup	3D Interface
Element Description	A family of 3D interface elements used for modelling delamination and crack propagation.
Number Of Nodes	6,8,12,16
Freedom	U, V, W: at each node.
Node Coordinates	X, Y, Z: at each node.

Geometric Properties

Not applicable (a zero thickness is assumed).

Material Properties

Linear	Not applicable
Matrix	Not applicable
Joint	Not applicable
Concrete	Not applicable
Elasto-Plastic	Not applicable

Creep	Not applicable	
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage	Not applicable	
Interface	Interface	MATERIAL PROPERTIES NONLINEAR 25
Rubber	Not applicable	
Generic Polymer	Not applicable	
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W: at each node.
Concentrated Loads	CL	Concentrated loads. Px, Py, Pz: at each node.
Element Loads	Not applicable.	
Distributed Loads	Not applicable.	
Body Forces	Not applicable.	
Velocities	VELO	Velocities. Vx, Vy, Vz: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay, Az: at nodes.
Initial Stress/Strains	Not applicable.	
Residual Stresses	Not applicable.	
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

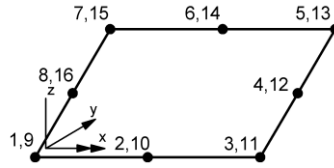
Solver	Stress (default): shear tractions in X and Y, and direct tractions. Strain: relative displacements in X, Y and Z directions.
Modeller	See Results Tables (Appendix K) .

Local Axes

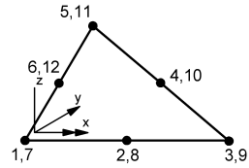
Element Name

Evaluated at each node.

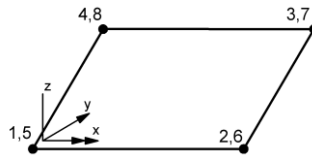
IS16



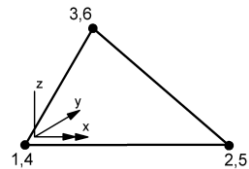
IS12



IS8



IS6



Evaluated at each node.

Sign Convention

A positive traction occurs if the local relative displacement (with respect to the first surface of the element) is a positive value, i.e. for the IS16 element at nodes $3 > 11$ the local relative displacement, E_z , would be positive if $(D_{z11} - D_{z3}) > 0$, where D_{zi} is the local displacement at node i .

Formulation

Geometric Nonlinearity

Total Lagrangian	Not applicable.
Updated Lagrangian	Not applicable.
Eulerian	Not applicable.
Co-rotational	Applicable to IS6 and IS8 elements.

Integration Schemes

Stiffness	Default.	3x3 (Newton-Cotes) (IS16), 2x2 (Newton Cotes) (IS8), 7-point cubic (IS12), 3-point (IS6)
	Fine.	As default

Mass Modelling

Not applicable.

Options

- 62 Continue solution if more than one negative pivot occurs.

- 229 Co-rotational geometric non-linearity.
- 252 Suppress pivot warning messages
- 261 Select the root with the lowest residual norm with arc-length.

Notes on Use

1. When defining the transient analysis control the arc-length procedure should be adopted with the option to select the root with the lowest residual norm [option 261].
2. It is recommended that fine integration [option 18] is selected for the parent elements.
3. The nonlinear convergence criteria should be selected to converge on the residual norm.
4. Option 62, Continue solution if more than one negative pivot occurs, should be selected to continue if more than one negative pivot is encountered and option 252 should be used to suppress pivot warning messages from the solution process.
5. The non-symmetric solver is selected automatically when mixed mode delamination is specified.
6. Although the solution is largely independent of the mesh discretisation, to avoid convergence difficulties it is recommended that at least 2 elements are placed in the process zone.

Restrictions

None.

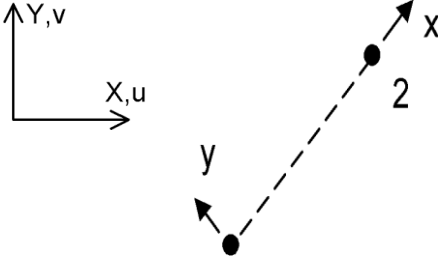
Recommendations on Use

These elements should be used at places of potential delamination between 3D continuum elements.

Chapter 11 : Non-Structural Mass Elements.

2D Point Mass Element

General

Element Name	PM2
	
Element Group	Non-Structural Mass
Element Subgroup	2D Point
Element Description	A 2D point mass element to model mass at a point.
Number Of Nodes	2. The 2 nd node is used to define the local x-axis.
Freedom	U, V: at each node.
Node Coordinates	X, Y: at each node.

Geometric Properties

Not applicable.

Material Properties

Linear	Not applicable	
Matrix	Not applicable	
Joint	Not applicable	
Mass	2D	MATERIAL PROPERTIES MASS 2 1
Concrete	Not applicable	
Elasto-Plastic	Not applicable	
Creep	Not applicable	
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage	Not applicable	
Rubber	Not applicable	
Generic Polymer	Not applicable	

Composite Not applicable
Field Not applicable

Loading

Prescribed Value CBF Constant body forces for element. Xcbf, Ycbf, Zcbf (applied as accelerations)

LUSAS Output

None

Local Axes

The 2nd node is used to define the local x-axis.

Sign Convention

☐ Not applicable.

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Not applicable.

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

105 Lumped mass matrix.

Notes on Use

1. Use to model point mass in a structure.

Restrictions

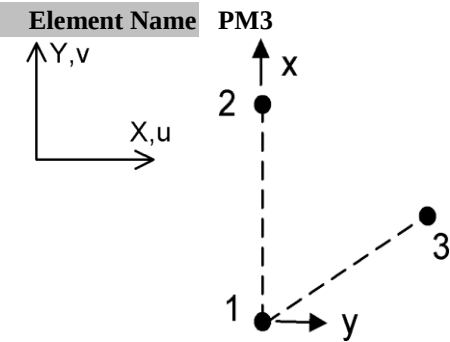
None.

Recommendations on Use

The 2D point mass element can be used to model point masses occur in a 2D structure.

3D Point Mass Element

General



Element Group	Non-Structural Mass
Element Subgroup	3D Point
Element Description	A 3D point mass element to model mass at a point.
Number Of Nodes	3. The 2nd node is used to define the local x-axis. The 2nd and 3rd node define the local x-y plane.
Freedom Node Coordinates	U, V, W: at each node. X, Y, Z: at each node.

Geometric Properties

Not applicable.

Material Properties

Linear	Not applicable	
Matrix	Not applicable	
Joint	Not applicable	
Mass	3D.	MATERIAL PROPERTIES MASS 3 1
Concrete	Not applicable	
Elasto-Plastic	Not applicable	
Creep	Not applicable	
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage	Not applicable	
Rubber	Not applicable	

Generic Polymer	Not applicable
Composite	Not applicable

Loading

Prescribed Value	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf (applied as accelerations)

Output

None

Local Axes

The 2nd node is used to define the local x-axis. The 2nd and 3rd node define the local x-y plane.

Sign Convention

☐ Not applicable.

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Not applicable.

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

105 Lumped mass matrix.

Notes on Use

1. Use to model point mass in a structure.

Restrictions

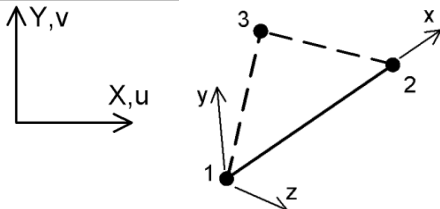
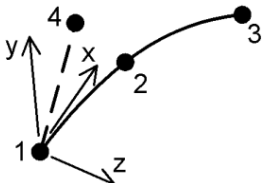
None.

Recommendations on Use

The 3D point mass element can be used to model point masses occur in a 3D structure.

3D Line Mass Elements

General

Element Name	LMS3	LMS4
		
Element Group	Non-Structural Mass	
Element Subgroup	3D Line	
Element Description	3D straight (LMS3) and curved (LMS4) line mass elements to model mass along an edge. The elements can accommodate varying mass along the length.	
Number Of Nodes	3 (LMS3). The 3 rd node is used to define the local x-y plane. 4 (LMS4). The 4 th node is used to define the local x-y plane.	
End Releases		
Freedoms	U, V, W: at each node.	
Node Coordinates	X, Y, Z : at each node.	

Geometric Properties

Not applicable.

Material Properties

Linear	Not applicable	
Matrix	Not applicable	
Joint	Not applicable.	
Mass	3D.	MATERIAL PROPERTIES MASS 3 2 (or 3)
Concrete	Not applicable	
Elasto-Plastic	Not applicable	
Creep	Not applicable	
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage	Not applicable	
Rubber	Not applicable	
Generic Polymer	Not applicable	

Composite Not applicable

Loading

Prescribed Value	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf (applied as accelerations)

Output

None

Local Axes

☐ [Standard Line Element](#)

Sign Convention

☐ Not applicable.

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Mass	Default.	2-point
	Fine	2-point (LMS2), 3-point (LMS3)

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

18	Invokes fine integration rule.
105	Lumped mass matrix.

Notes on Use

1. Use to model mass on an edge in a structure.

Restrictions

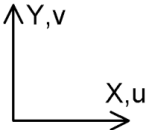
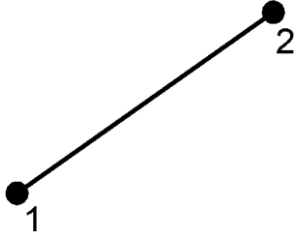
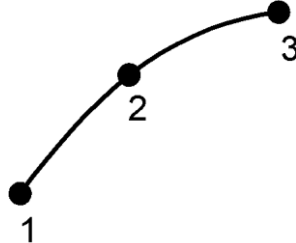
- ☐ [Ensure mid-side node centrality](#)
- ☐ Avoid excessive element curvature

Recommendations on Use

3D line mass elements can be used to model masses along an edge in a 3D structure.

2D Line Mass Elements

General

Element Name	LM2	LM3
		
Element Group	Non-Structural Mass	
Element Subgroup	2D Line	
Element Description	2D straight (LM2) and curved (LM3) line mass elements to model mass along an edge. The elements can accommodate varying mass along the length.	
Number Of Nodes	2 (LM2). 3 (LM3).	
End Releases		
Freedom	U, V: at each node.	
Node Coordinates	X, Y: at each node.	

Geometric Properties

Not applicable.

Material Properties

Linear	Not applicable	
Matrix	Not applicable	
Joint	Not applicable	
Mass	2D.	MATERIAL PROPERTIES MASS 2 2 (or 3)
Concrete	Not applicable	
Elasto-Plastic	Not applicable	
Creep	Not applicable	
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage	Not applicable	

Rubber	Not applicable
Generic Polymer	Not applicable
Composite	Not applicable

Loading

Prescribed Value	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf (applied as accelerations)

Output

None

Local Axes

- ☐ [Standard Line Element](#)

Sign Convention

- ☐ Not applicable.

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Mass	Default.	2-point
	Fine	2-point (LM2), 3-point (LM3)

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

18	Invokes fine integration rule.
105	Lumped mass matrix.

Notes on Use

1. Use to model mass on an edge in a structure.

Restrictions

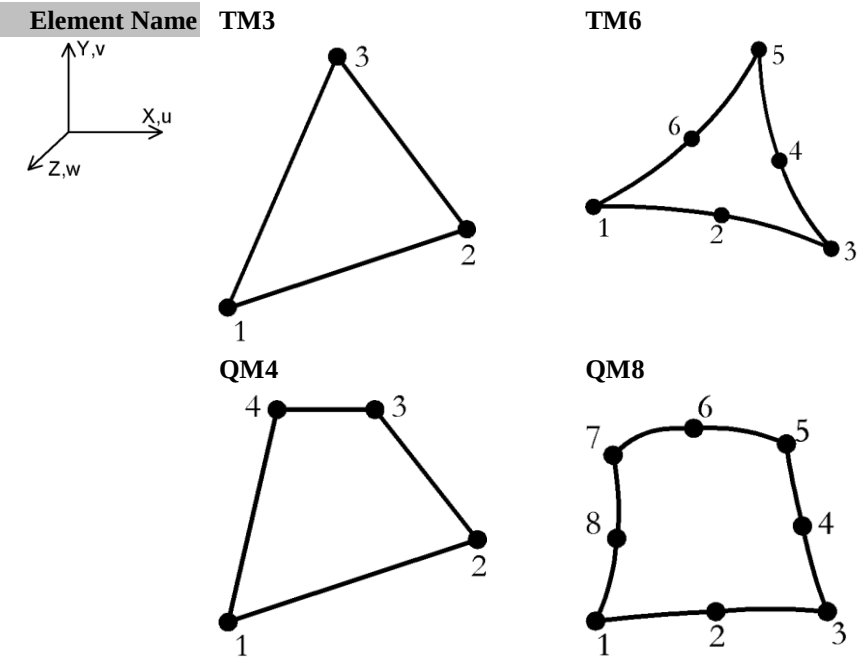
- ❑ [Ensure mid-side node centrality](#)
- ❑ Avoid excessive element curvature

Recommendations on Use

2D line mass elements can be used to model masses along an edge in a 2D structure.

Surface Mass Elements

General



Element Group	Non-Structural Mass
Element Subgroup	3D Surface
Element Description	3D surface mass elements to model mass on a surface.
Number Of Nodes	3,4,6 or 8.
End Releases	
Freedoms	U, V, W: at each node.
Node Coordinates	X, Y, Z : at each node.

Geometric Properties

Not applicable.

Material Properties

Linear	Not applicable	
Matrix	Not applicable	
Joint	Not applicable	
Mass	3D	MATERIAL PROPERTIES MASS 3 (3,4,6 or 8)
Concrete	Not applicable.	
Elasto-Plastic	Not applicable.	
Creep	Not applicable	
Damage	Not applicable	
Viscoelastic	Not applicable	
Shrinkage	Not applicable	
Rubber	Not applicable	
Generic Polymer	Not applicable	
Composite	Not applicable.	

Loading

Prescribed Value	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf (applied as accelerations)

Output

None

Local Axes

☐ [Standard Surface Element](#)

Sign Convention

Not applicable.

Formulation

Geometric Nonlinearity

Not applicable.

Integration Schemes

Mass	Default.	1-point (TM3), 3-point (TM6), 4-point (QM4,QM8)
	Fine	3-point (TM3, TM6), 4-point (QM4), 9-point (QM8)

Mass Modelling

- ☐ Consistent mass (default).
- ☐ Lumped mass.

Options

- 18 Invokes fine integration rule.
- 105 Lumped mass matrix.

Notes on Use

1. Use to model mass on a surface in a structure.

Restrictions

- ☐ [Ensure mid-side node centrality](#)
- ☐ [Avoid excessive element curvature](#)
- ☐ Avoid excessive aspect ratio

Recommendations on Use

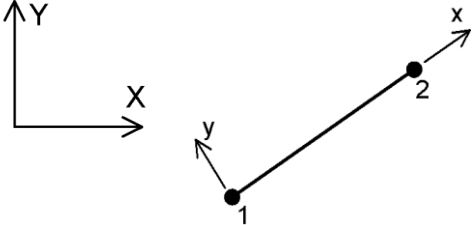
The surface mass elements can be used to model masses on a surface 3D structures.

Chapter 12 : Rigid Elements.

Rigid Surface 2D Elements

General

Element Name	R2D2



Element Group	Rigid
Element Subgroup	2D Rigid Surface
Element Description	2D Rigid Surface elements capable of modelling non-deformable surfaces in a contact analysis.
Number Of Nodes	2
Freedom	U, V at each node
Node Coordinates	X, Y at each node.

Geometric Properties

Not applicable.

Material Properties

Linear Isotropic: MATERIAL PROPERTIES (Elastic: Isotropic)

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V at each node.
Concentrated Loads	Not applicable.	
Element Loads	Not applicable.	
Distributed Loads	Not applicable.	
Body Forces	Not applicable.	
Velocities	VELO	Velocities. Vx, Vy at nodes.
Accelerations	ACCE	Acceleration Ax, Ay at nodes.
Initial Stress/Strains	Not applicable.	
Residual Stresses	Not applicable.	

Temperatures	Not applicable.
Field Loads	Not applicable.
Temp Dependent Loads	Not applicable.

LUSAS Output

Solver	Displacements & Reactions only.
Modeller	Displacements & Reactions only.

Formulation

Geometric Nonlinearity

Total Lagrangian	Depends on the other surface (deformable surface) which is in contact with the rigid surface. See the related section for the deformable surface elements.
Updated Lagrangian	As above.
Eulerian	As above.
Co-rotational	As above.

Integration Schemes

Not applicable.

Mass Modelling

Not applicable.

Restrictions

- A rigid surface cannot contact another rigid surface.
- Rigid surface elements do not accept external applied forces.

Notes on use

1. All the rigid surface element nodes must be fully restrained.
2. There is no stress and strain calculation for these elements.
3. If rigid slideline surfaces are defined there is no need to assign geometric and material properties to these elements. However, when using automatic contact surfaces, linear elastic isotropic material properties need to be assigned.

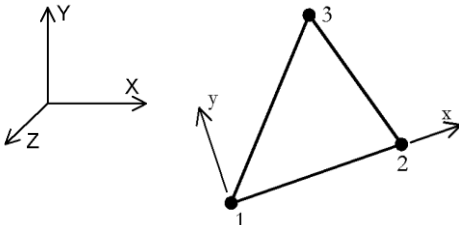
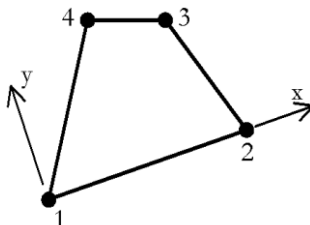
4. For saving analysis time a one pass contact algorithm can be used. In this case only the penetration of the deformable surface into the rigid surface is checked. To avoid the penetration of the rigid surface into the deformable surface use either the default two pass algorithm or a finer mesh on the deformable surface.

Recommendations on Use

These elements should be used when one of the surfaces which come into contact is non-deformable. Using these elements will make the analysis faster.

Rigid Surface 3D Elements

General

Element Name	R3D3	R3D4
		
Element Group	Rigid	
Element Subgroup	3D Rigid Surface	
Element Description	3D Rigid Surface elements capable of modelling non-deformable surfaces in a contact analysis.	
Number Of Nodes	3/4	
Freedom Node	U, V, W at each node.	
Coordinates	X, Y, Z at each node.	

Geometric Properties

Not applicable.

Material Properties

Linear Isotropic: MATERIAL PROPERTIES (Elastic: Isotropic)

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W at each node.
Concentrated Loads	Not applicable.	
Element Loads	Not applicable.	
Distributed Loads	Not applicable.	
Body Forces	Not applicable.	
Velocities	VELO	Velocities. Vx, Vy, Vz at nodes.
Accelerations	ACCE	Acceleration Ax, Ay, Az at nodes.
Initial Stress/Strains	Not applicable.	
Residual Stresses	Not applicable.	

Temperatures	Not applicable.
Field Loads	Not applicable.
Temp Dependent Loads	Not applicable.

LUSAS Output

Solver	Displacements & Reactions only.
Modeller	Displacements & Reactions only.

Formulation

Geometric Nonlinearity

Total Lagrangian	Depends on the other surface (deformable surface) which is in contact with the rigid surface. See the related section for the deformable surface elements.
Updated Lagrangian	As above.
Eulerian	As above.
Co-rotational	As above.

Integration Schemes

Not applicable.

Mass Modelling

Not applicable.

Restrictions

- A rigid surface cannot contact another rigid surface.
- Rigid surface elements do not accept external applied forces.

Notes on use

1. All the rigid surface element nodes must be fully restrained.
2. There is no stress and strain calculation for these elements.
3. If rigid slideline surfaces are defined there is no need to assign geometric and material properties to these elements. However, when using automatic contact surfaces, linear elastic isotropic material properties need to be assigned.

4. For saving analysis time a one pass contact algorithm can be used. In this case only the penetration of the deformable surface into the rigid surface is checked. To avoid the penetration of the rigid surface into the deformable surface use either the default two pass algorithm or a finer mesh on the deformable surface.

Recommendations on Use

These elements should be used when one of the surfaces which come into contact is non-deformable. Using these elements will make the analysis faster.

Appendix A : Element and Pressure Loads.

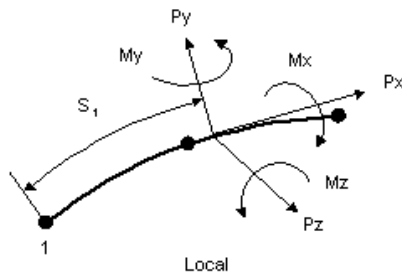
ELDS Element Loads

These are referred to as Internal Beam Point Loads and Internal Beam Distributed Loads within LUSAS Modeller.

Parameter	Description
ltype	Element load type
S1, S2	Distances to specified loads
Px, Py, Pz	Point loads in local/global directions
Mx, My, Mz	Point moments in local/global directions
Wx, Wy, Wz	Distributed loads in local/global directions

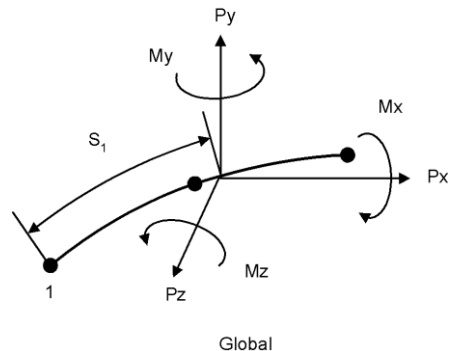
ltype 11

Point loads and moments in local directions



ltype 12

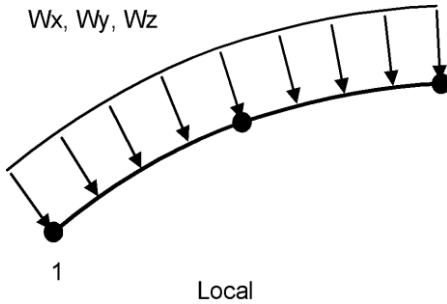
Point loads and moments in global directions



Itype 21

Uniformly distributed loads in local directions

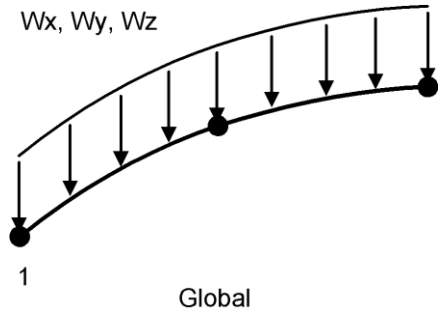
W_x, W_y, W_z



Itype 22

Uniformly distributed loads in global directions

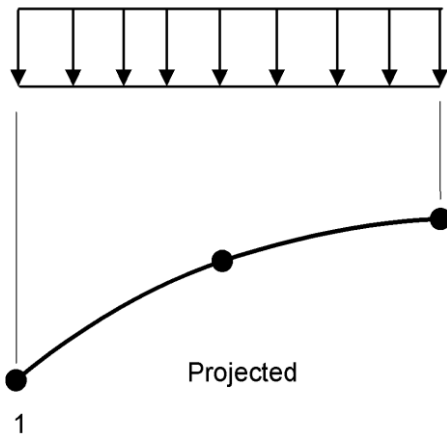
W_x, W_y, W_z



Itype 23

Uniformly distributed projected loads in global directions

W_x, W_y, W_z

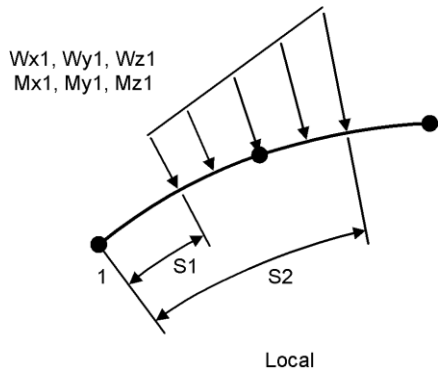


Itype 31

Distributed loads in local directions. Multiple load sets supported.

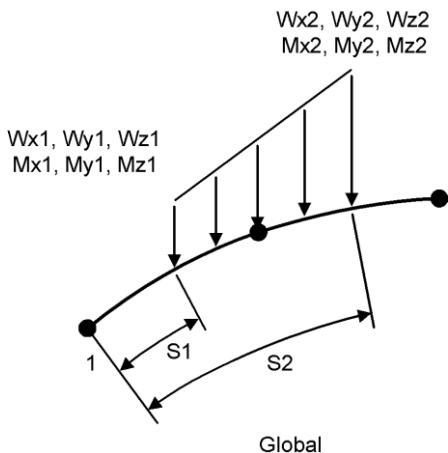
W_{x2}, W_{y2}, W_{z2}
 M_{x2}, M_{y2}, M_{z2}

W_{x1}, W_{y1}, W_{z1}
 M_{x1}, M_{y1}, M_{z1}



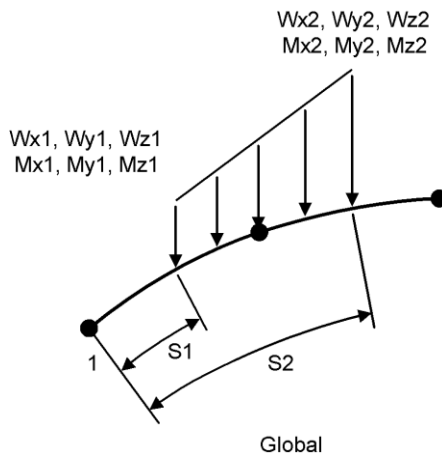
Itype 32

Distributed loads in global directions. Multiple load sets supported.



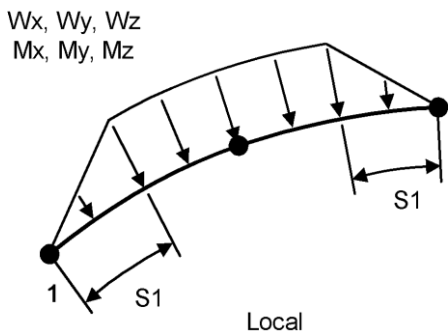
Itype 33

Distributed projected loads in global directions. Multiple load sets supported.



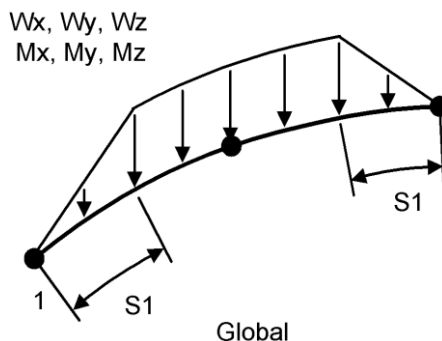
Itype 41

Trapezoidal loads in local directions
Definition only supported in LUSAS Solver. In LUSAS Modeller trapezoidal beam loads are defined in accordance with Itype 31.



Itype 42

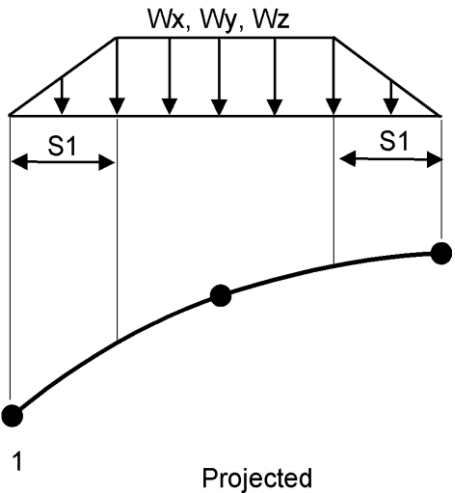
Trapezoidal loads in global directions
Definition only supported in LUSAS Solver. In LUSAS Modeller trapezoidal beam loads are defined in accordance with Itype 32.



Ittype 43

Trapezoidal projected loads in global directions

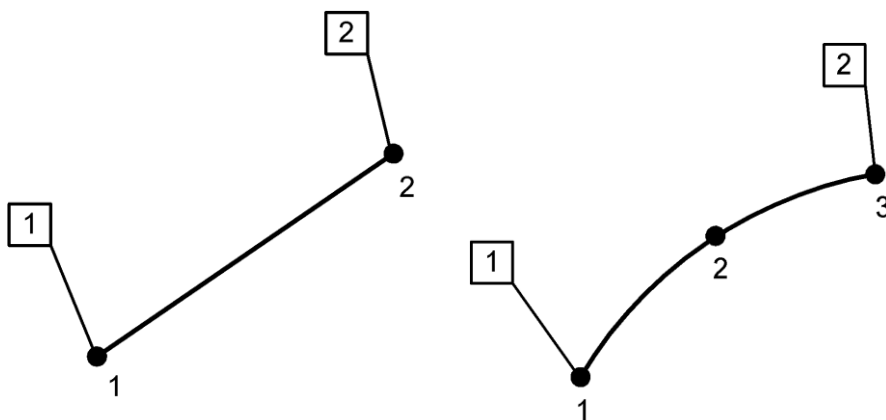
Definition only supported in LUSAS Solver. In LUSAS Modeller trapezoidal beam loads are defined in accordance with Ittype 33.



ENVT/TDET Environmental Temperature Loading

Contains some or all of:

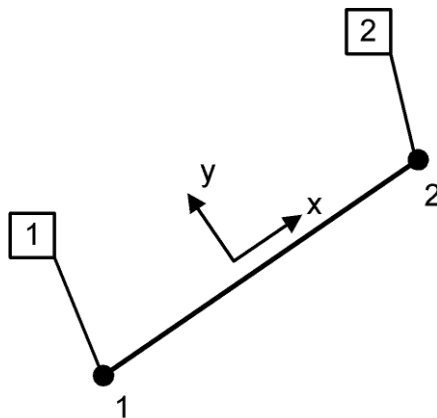
<u>Parameter</u>	<u>Description</u>
je	External environmental temperature.
hc	Convective heat transfer coefficient.
hr	Radiative heat transfer coefficient.
T	Temperature for element.



Face Numbering Convention for Thermal Bars

Note: The environmental temperature loading for node 2 cannot be specified for a 3 noded bar.

FLD Face loading applied to thermal bars

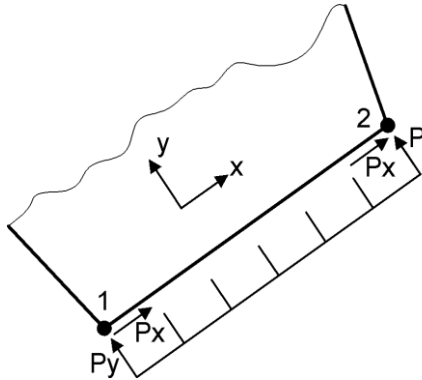


Face number = local node number
Face Numbering Convention for Thermal Bars

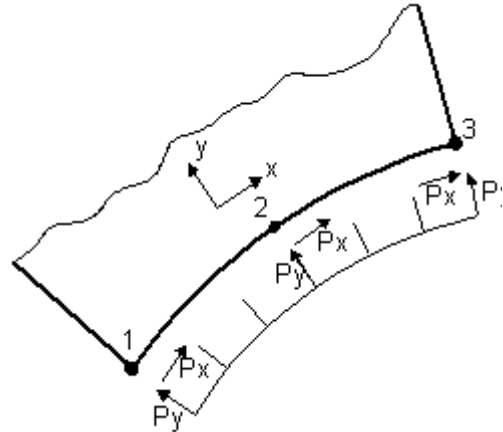
Face Loads On 2D Continuum Elements

Parameter	Description
P_x, P_y	Face pressures defined at nodes in local x, y directions

2-Noded Element Faces



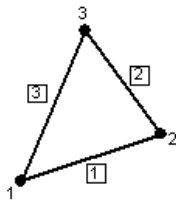
3-Noded Element Faces



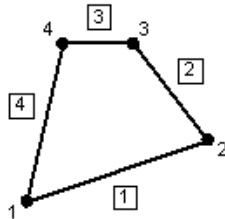
Note: Face loads for explicit dynamics elements are constant, i.e. the average of the input nodal pressures.

Face Numbering Convention

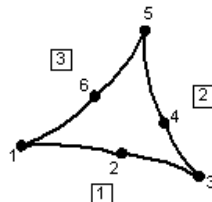
3 Noded Elements



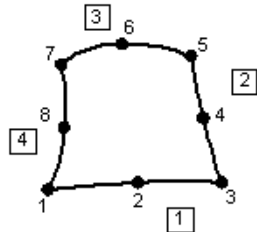
4 Noded Elements



6 Noded Elements



8 Noded Elements

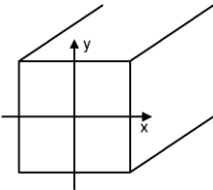


Face Loads On 3D Continuum Elements

Parameter	Description
Px, Py, Pz	Face pressures defined at nodes in local x, y directions acting positively in the local coordinate directions

Note: Face loads for explicit dynamics elements are constant, i.e. the average of the input nodal pressures.

Local Face Coordinates

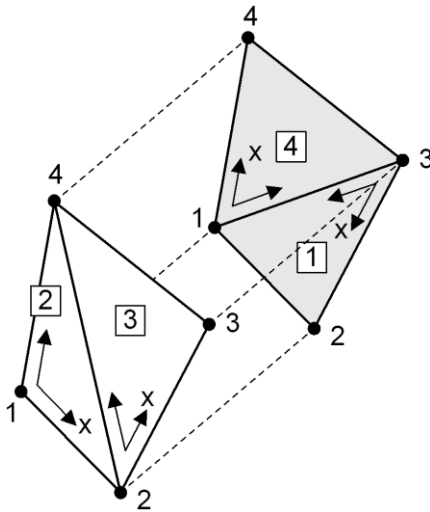


Face Numbering Convention

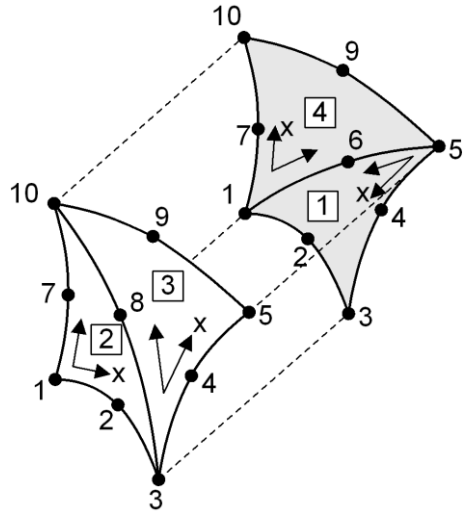
The following diagrams show exploded view of the various 3D elements. The grey faces show the element external faces that can be seen from a single perspective point, the white faces depict the internal faces from the same view point.

Note: The views of the internal faces show the x-axis direction from the inside. Take care when converting this to a view from the outside of the element.

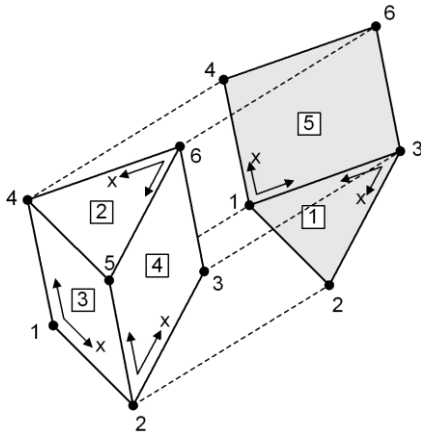
4-Noded Tetrahedra



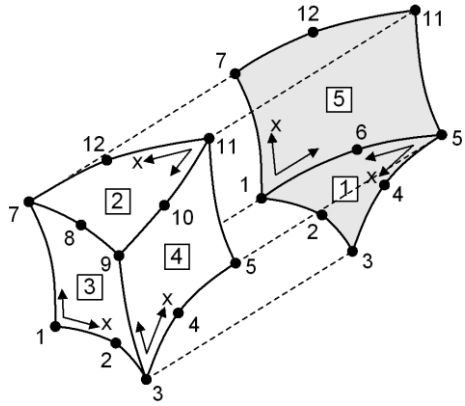
10-Noded Tetrahedra



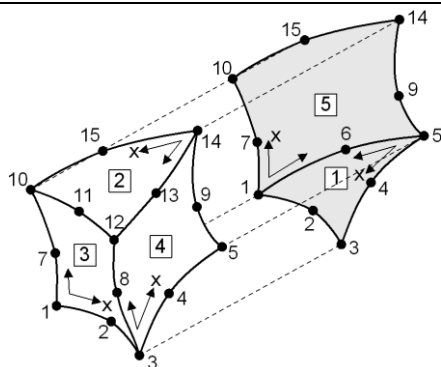
6-Noded Pentahedra



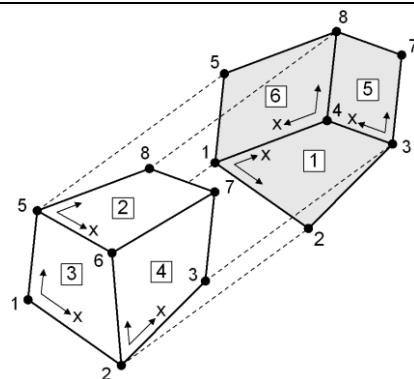
12-Noded Pentahedra



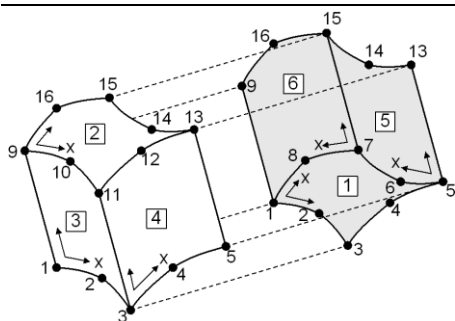
15-Node Pentahedra



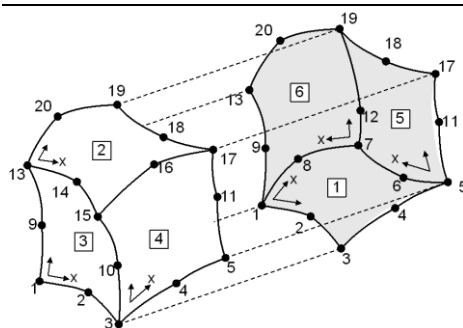
8-Noded Hexahedra



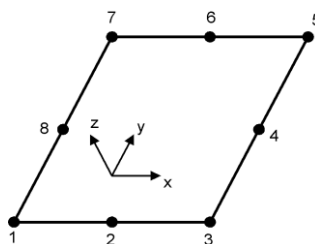
16-Noded Hexahedra



20-Noded Hexahedra



UDL Loads on Shells



Appendix B : Element Restrictions.

Mid-side Node Centrality

The mid-length node must be equidistant from the end nodes. Mid-side nodes may be automatically corrected for elements with global translational mid-side node freedoms using Option 49. The mid-side node is moved along the existing element edge until it is positioned centrally.

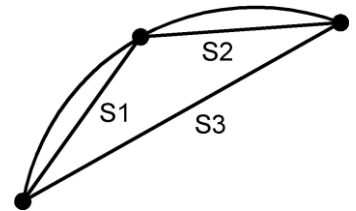
Excessive Element Curvature

Elements must not be excessively curved. A warning will be invoked (but the analysis will continue) if the element curvature is not in accordance with the following inequalities:

i) $ABS (S1-S2) / (S1+S2) < 0.05$

ii) $(S1+S2) / S3 < 1.02$

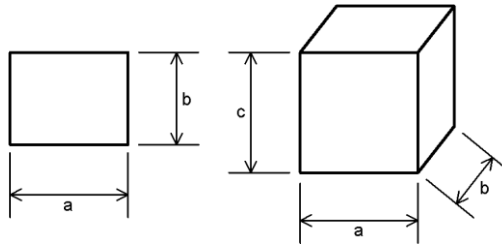
Where the function ABS returns the absolute value of the arguments.



Excessive Aspect Ratios

An aspect ratio can be defined as the ratio of the longest to shortest element side lengths, such that:

- ☐ $R = \max (a/b, b/a)$ for surface elements (e.g. 2D continuum, plates and shells)
- ☐ $R = \max (a/b, b/a, c/a, c/b, \dots)$ for three dimensional solid elements



Elements must not have an excessive aspect ratio. A warning will be invoked (but the analysis will continue) if the element aspect ratio is greater than 10.

In general, severe distortion of an element will affect the accuracy of the stress distribution through an element. The type of stress field being imposed is also of importance, since a badly shaped element will still yield a good distribution in the presence of a constant uniaxial stress field, but not when subjected to a full stress field in which any of the components have a significant variation across the element.

The force equilibrium for the element will always be satisfied.

Excessive Warping

The four nodal points defining quadrilateral surface elements should be coplanar. However a small out of plane tolerance is permitted to allow a slightly warped shape according to

$$z < 0.01(L_{12})$$

where z is the out of plane distance of a node,

and L_{12} is the length between the first and second nodes.

If the above inequality is exceeded a warning will be issued but the analysis will proceed.

Appendix C : Local Element Axes.

Standard Joint Element

Local x-axis The local x-axis is defined by the vector between the first and the third nodes of the element topology.

Note: *The third node must be different from nodes 1 and 2 of the topology.*

Standard Line Element

Local x axis The local x-axis lies along the element in the direction in which the element nodes are defined. For curved elements the local x-axis is the tangent to the curve.

Local y axis The local xy plane is either defined by a dummy node and the two end nodes, or (in the absence of a dummy node), defined by the two end nodes and the central node. For the latter case, the local y-axis is perpendicular to the x-axis and on the positive convex side.

Local z axis The local z-axis forms a right-handed set with the local xy plane.

For cross-section beams the top surface is defined by the local +ve z direction.

Note: *Default line axes are defined in Modeller with the local x axis of the element following the line direction. The element local z is then defined in the XZ plane unless the local x axis is aligned to the global Z axis in which case the element local z axis is aligned with the global Y axis.*

Standard Surface Element

Local x axis For 3 or 4 noded elements the local x-axis is defined by a line joining the first and second element nodes. For 6 and 8 noded elements the local x-axis is the tangent to the curve between the first 3 nodes.

Local y axis The local xy-plane is defined by the remaining nodes, the local y-axis being perpendicular to the x-axis and forming a right-handed set with the x-axis and the xy plane.

Local z-axis The local z-axis forms a right-handed set with the local x and y-axes.

For shell elements the top surface is defined by the local +ve z direction.

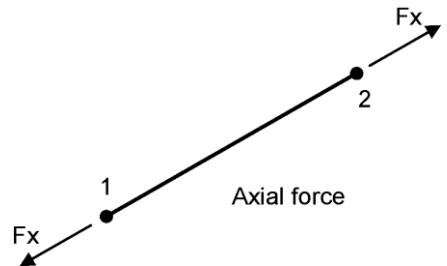
Appendix D : Sign Conventions.

Standard Bar Element

Axial force

(+ve) Axial tension

(-ve) Axial compression



Standard Beam Element

Numerically Integrated Beam Elements

Axial force

(+ve) Axial tension

(-ve) Axial compression

Bending Moment

(+ve) Hogging moment (Top of beam in tension)

(-ve) Sagging moment (Bottom of beam in tension)

Note: The top/bottom of the beam are determine by the element axes.

Torsion

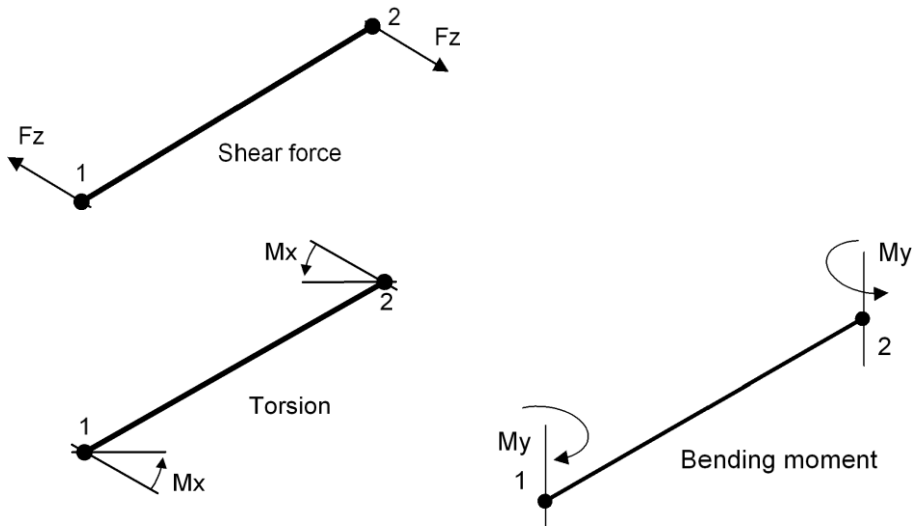
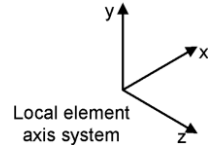
(+ve) Anti-clockwise rotation (1st node), clockwise rotation (3rd node)

(-ve) Clockwise rotation (1st node), anti-clockwise rotation (3rd node)

Grillage Elements

End Forces and Rotations

Positive end forces and rotations for grillage elements are those acting on the element nodes in local directions, and are as follows:



Note that when a reference path has been specified, additional force/moment components are available, and for this situation the x, y, and z element axes relate to longitudinal, transverse and vertical terms respectively. For instance M_y will relate to MF (longitudinal) - the flexural moment in longitudinal members that are following the path and MF (transverse) - the flexural moment in the transverse members that are orthogonal or skewed in relation to the reference path. Similarly, F_z will relate to FV (longitudinal) - the force in the vertical direction for longitudinal members that are following the path and FV (transverse) - the vertical direction for transverse members that are orthogonal or skewed in relation to the reference path.

Internal forces

These forces follow the sign convention for numerically integrated beams.

Axial force	Bending Moment	Torsion
Not applicable	(+ve) Sagging moment	(+ve) Anti-Clockwise rotation (end 1)
	(-ve) Hogging moment	(-ve) Clockwise rotation (end 2)

Sign convention in Modeller for bending moment

(+ve) Top of beam in tension

(-ve) Bottom of beam in tension

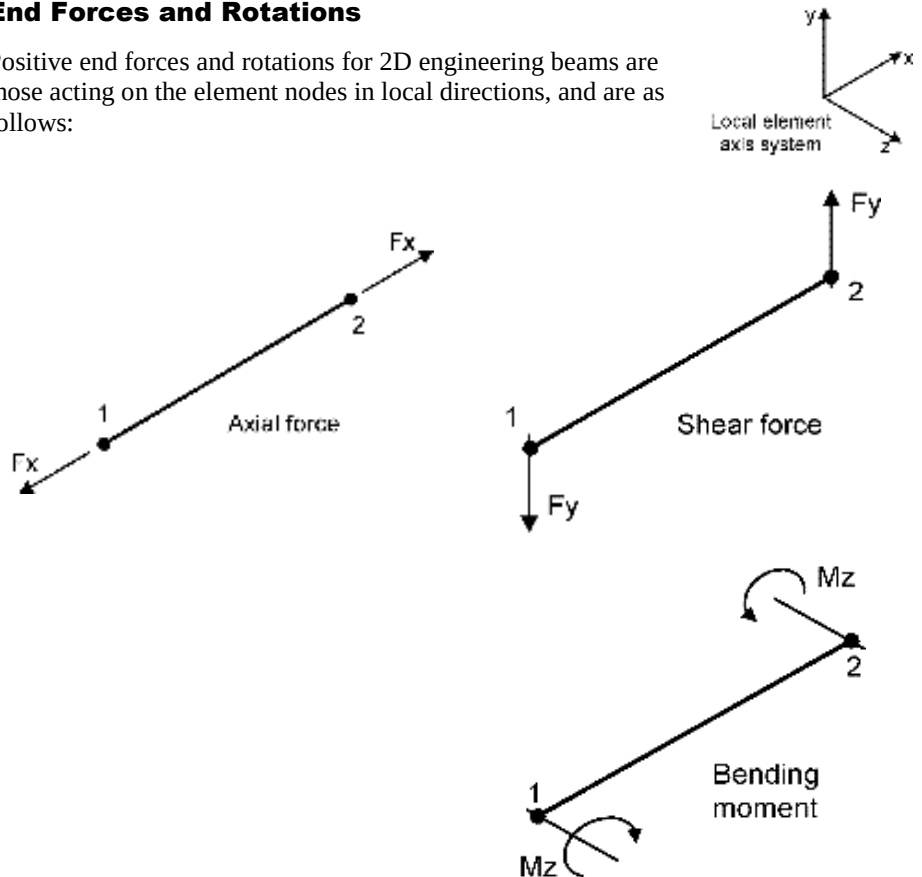
Where the top/bottom of the beam are determined by the element axes

See [numerically integrated beam sign convention](#).

2D Engineering Beam Elements

End Forces and Rotations

Positive end forces and rotations for 2D engineering beams are those acting on the element nodes in local directions, and are as follows:



Internal forces

These forces follow the sign convention for numerically integrated beams.

<u>Axial force</u>	<u>Bending Moment</u>	<u>Torsion</u>
(+ve) Axial tension	(+ve) Hogging moment	(+ve) Anti-Clockwise rotation (end 1)
(-ve) Axial compression	(-ve) Sagging moment	(-ve) Clockwise rotation (end 2)

Sign convention in Modeller for bending moment

(+ve) Top of beam in tension

(-ve) Bottom of beam in tension

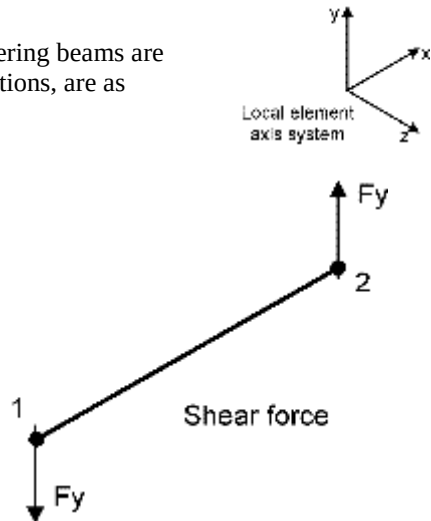
Where the top/bottom of the beam are determined by the element axes

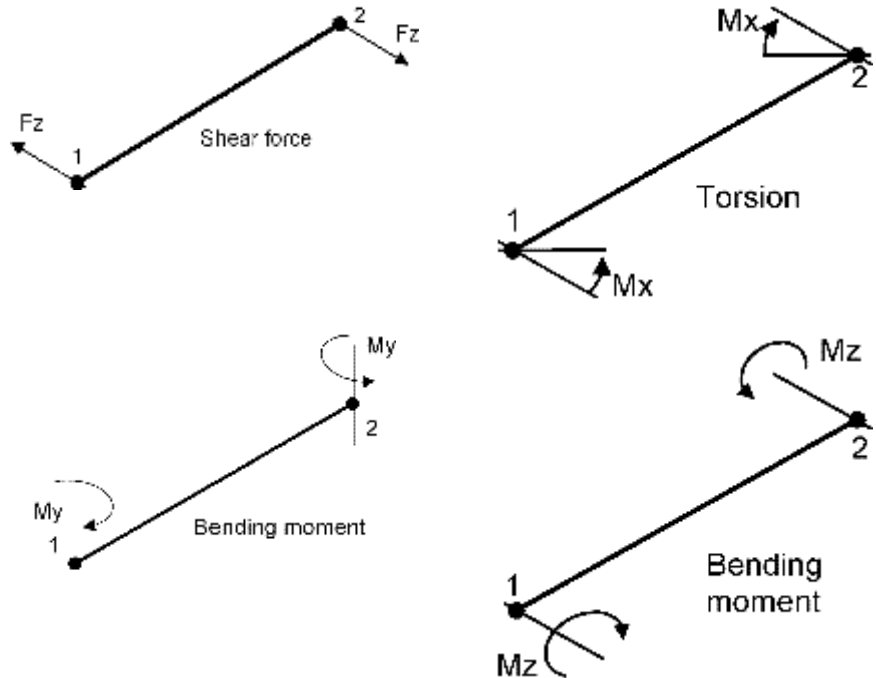
See [numerically integrated beam sign convention](#).

3D Engineering Beam Elements

End Forces and Rotations

Positive end forces and rotations for 3D engineering beams are those acting on the element nodes in local directions, are as follows:





Internal forces

These forces follow the sign convention for numerically integrated beams.

Axial force	Bending Moment	Torsion
(+ve) Axial tension	(+ve) Hogging moment	(+ve) Anti-Clockwise rotation (end 1)
(-ve) Axial compression	(-ve) Sagging moment	(-ve) Clockwise rotation (end 2)

Sign convention in Modeller for bending moment

- (+ve) Top of beam in tension
- (-ve) Bottom of beam in tension

Where the top/bottom of the beam are determined by the element axes

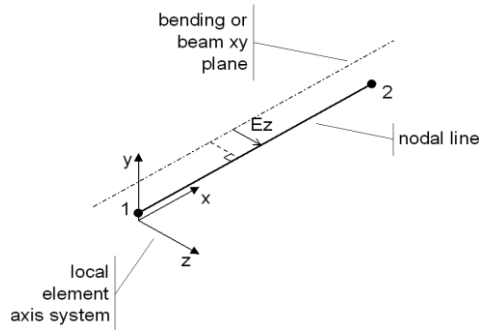
See [numerically integrated beam sign convention](#).

Standard Beam Eccentricities

Eccentricities are optional geometric properties for some elements and may be specified if the nodal line of the element does not lie along the required bending line/plane for the structural component being modelled.

Measurement of E_z (see diagram) is **from** the required bending plane (the beam xy plane) **to** the nodal line in the local element axis z-direction. If a beam xy plane is required such that it has negative local z coordinates relative to the nodal line, the eccentricity is positive.

Similarly, measurement of E_y is **from** the required bending plane (the beam xz plane) **to** the nodal line in the local element axis y-direction. If a beam xz plane is required such that it has negative local y coordinates relative to the nodal line, the eccentricity is positive.



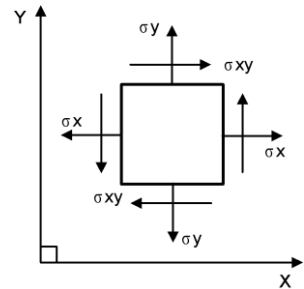
Standard 2D Continuum Element

Direct stress

(+ve) Tension
(-ve) Compression

Shear stress

(+ve) Shear into XY quadrant
(-ve) Shear into XY quadrant



Note. Positive stress values are shown.

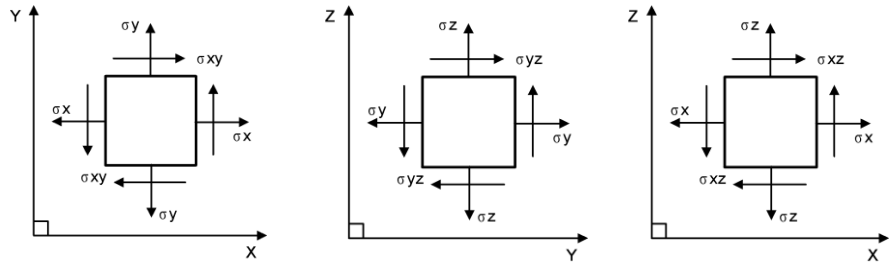
Standard 3D Continuum Element

Direct stress

(+ve) Tension
(-ve) Compression

Shear stress

(+ve) Shear into XY, YZ and XZ quadrants
(-ve) Shear into XY, YZ and XZ quadrants



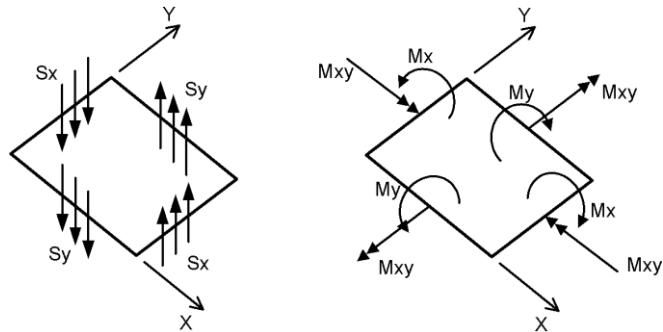
Note. Positive stress values shown.

Standard Plate Element

Flexural stress

(+ve) Hogging moment (producing +ve stresses on the element top surface)

(-ve) Sagging moment (producing -ve stresses on the element top surface)



The +ve local z-direction defines the top surface.

Thin Shell Element

Membrane stress

(+ve) Direct tension

(-ve) Direct compression

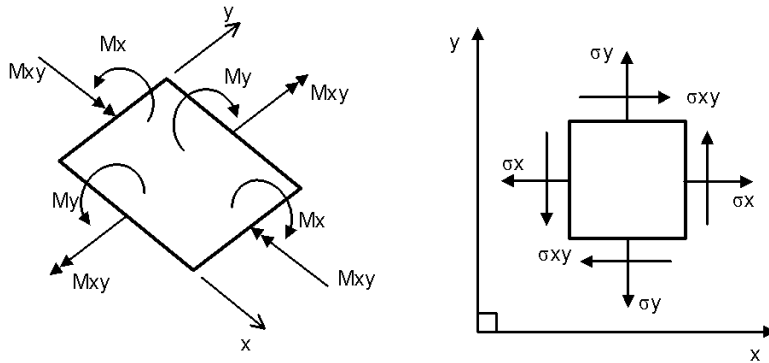
(+ve) In-plane shear into xy quadrant

(-ve) In-plane shear into xy quadrant

Flexural stress

(+ve) Hogging moment (producing +ve stresses on the element top surface)

(-ve) Sagging moment (producing -ve stresses on the element top surface)

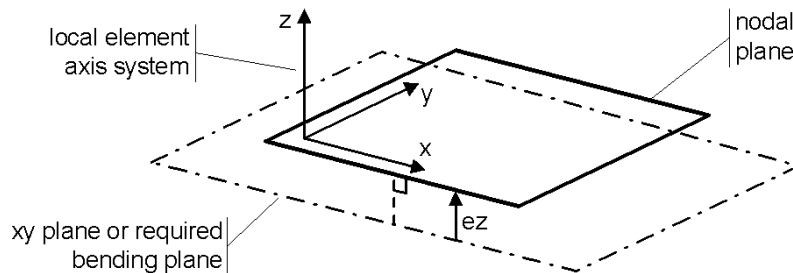


Notes

- Positive stress values are shown.
- The +ve local z-direction defines the top surface.

Thin Shell Eccentricity

Eccentricity is an optional geometric property for this element type and may be specified if the nodal plane of the element does not lie along the required bending plane for the structural component being modelled.

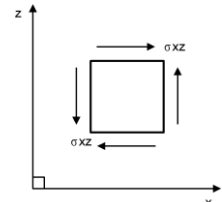
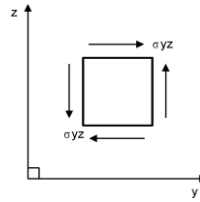
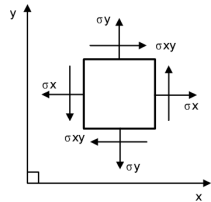


Measurement of ez is **from** the required bending plane **to** the nodal plane in the local element axis z-direction.

Thick Shell Element

Continuum Stress

Direct stress	(+ve)	Tension
	(-ve)	Compression
Shear stress	(+ve)	Shear into xy, yz and zx quadrants
	(-ve)	Shear into xy, yz and zx quadrants

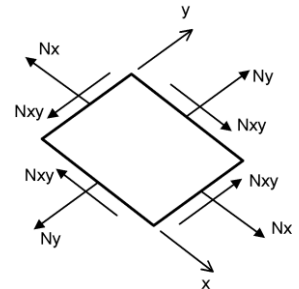
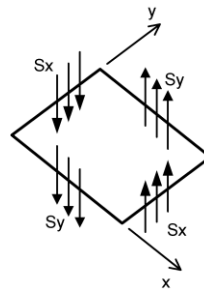
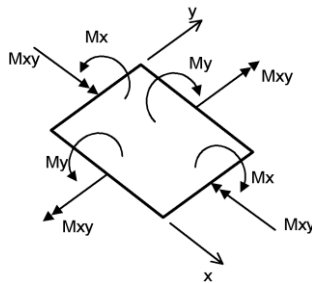


Stress Resultant

Membrane stress (+ve) Direct tension
(-ve) Direct compression

(+ve) In-plane shear into xy quadrant
(-ve) In-plane shear into xy quadrant

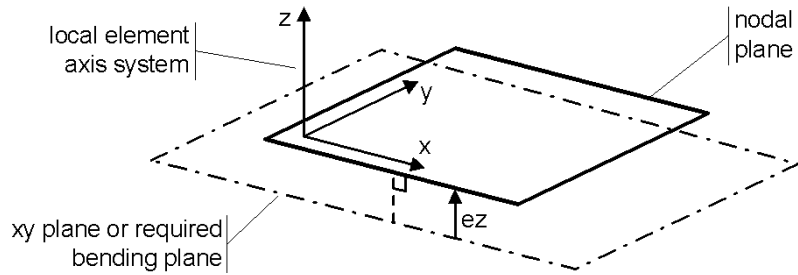
Flexural stress (+ve) Hogging moment (producing +ve stresses on the element top surface)
(-ve) Sagging moment (producing -ve stresses on the element top surface)



The +ve local z-direction defines the top surface.

Thick Shell Eccentricity

Eccentricity is an optional geometric property for this element type and may be specified if the nodal plane of the element does not lie along the required bending plane for the structural component being modelled.

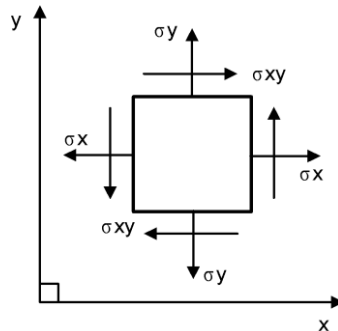


Measurement of ez is **from** the required bending plane **to** the nodal plane in the local element axis z-direction.

Standard Membrane Element

Direct stress (+ve) Tension
(-ve) Compression

Shear stress (+ve) Shear into xy quadrant
(-ve) Shear into xy quadrant



Standard Field Element

Potential

(+ve) +ve field value, dT/dx rate of change of field in x direction

Standard Joint Element

Direct force : (+ve) Tension and (-ve) Compression

Spring Moment : (+ve) for positive rotational spring strain and (-ve) for negative rotational spring strain

Appendix E : Thick Shell Notation.

Thick Shell Nodal Rotation

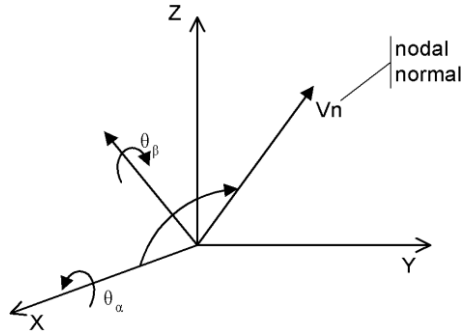
Problems with Singularities

In general, five degrees of freedom will be associated with each shell node: three translations and two rotations. The first axis of rotation will be defined by one of the global axes. The second axis of rotation is defined by the vector product of the selected global axis and the nodal normal.

Choosing one global axis to define the first rotation is not possible for all cases as singularities can occur depending on the orientation of the shell. As the topology of the shell cannot be known a means of choosing suitable rotations after the shell orientation has been defined must be provided.

How the Nodal Systems are Defined

The axis defining the θ_α rotation is chosen by examining the global components of the nodal normal. The smallest (absolute) component of the normal vector defines the global axis to be chosen as the first axis of rotation. The vector product of this axis and the nodal normal defines the axis for the second rotation θ_β . If the nodal normal coincides with the global Z axis, the global X axis will be chosen to define θ_α . In this instance, the X and Y components will



both be minimum values. When two components define the same minimum value the order of priority for selection of the axis is X, Y, Z. Note that, in general, the axes of rotation and the nodal normal will form a non-orthogonal left-handed set. The rotations are indicated in the following figure where the global x axis has been used to define θ_α :

Five or Six Degrees of Freedom at a Node

LUSAS Solver will automatically select five degrees of freedom at a node, with rotations defined as above, unless:

- ☐ The maximum angle between the normals of adjacent elements meeting at the node is greater than 20 degrees. The value of 20 degrees is selected by default and may be changed using the SYSTEM parameter SHLANG.
- ☐ Beam, joint or other shell element types are connected to the node
- ☐ [Concentrated loads](#) or [support conditions](#) have been specified at the node using LUSAS Modeller
- ☐ Option 278 has been specified
- ☐ Six degrees of freedom have been selected for the node within the NODAL FREEDOMS data chapter. If six degrees of freedom are used at a node the rotations will relate to the global axes, θ_X , θ_Y and θ_Z unless TRANSFORMED FREEDOMS have been specified. It is recommended that the default value for SHLANG is retained wherever possible.

When are Six Degrees of Freedom Necessary?

Rotations relating to global axes will be required in the following circumstances:

- ☐ When a branched shell connection exists in the structure to be analysed. LUSAS Solver will automatically detect this and assign six degrees of freedom to nodes along the branch connection.

- ❑ When connecting with other element types. Six degrees of freedom will automatically be assigned to shell nodes connected to beams, joints or other shell element types.
- ❑ When boundary conditions or loading cannot be easily specified using the above definition of rotations, e.g. when applying moments or using symmetry.

If the rotations $\theta_\alpha, \theta_\beta$ will not allow the required loading or symmetry conditions to be applied, rotations about global axes may be enforced using NODAL FREEDOMS. The use of TRANSFORMED FREEDOMS will then allow the rotations to be related to a more convenient local orthogonal set if necessary. If six degrees of freedom at a node are enforced using NODAL FREEDOMS (i.e. not set automatically by LUSAS Solver) singularities may occur if the **in-plane rotation** (about the normal) is **not restrained**.

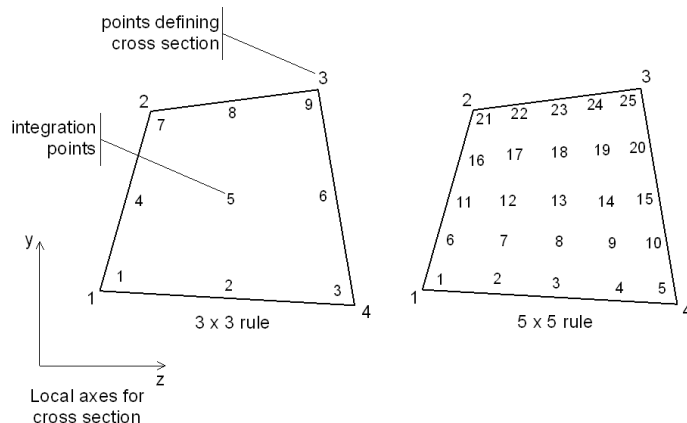
Appendix F :

Newton Coates

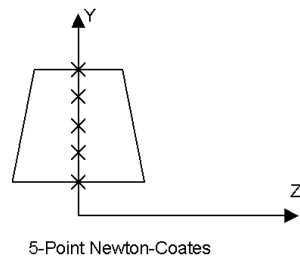
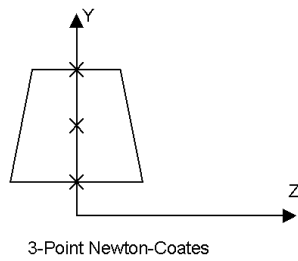
Integration.

Newton-Cotes Integration Points

For beam elements BMX3, BSX4 and BXL4 the rigidity is computed by integration of the cross section. The default integration employs a 3x3 Newton Cotes rule for linear materials and a 5x5 rule for nonlinear materials. These may be altered by the user within the GEOMETRIC PROPERTIES definition. The locations of the default integration points are shown in the accompanying diagram, together with the local axes for the beam cross section (note the different corner numbering). The integration points are equally spaced along a particular natural ordinate for the section. The integration point numbers shown correspond with those given in the stress output for the element. More information on the cross sectional integration for these elements is available in the *LUSAS Theory Manual*.



Newton-Cotes Integration Points for 3D Elements



Newton-Cotes Integration Points for 2D Elements

Appendix G : Shear Area and Torsional Constant.

Shear Areas

In beams of small span to depth ratio, the shear stresses are likely to be high and the resulting deflection due to shear may not be negligible. The shear area is used to control the amount of shear deformation which will occur (A_{sz} , A_{sy}). For various sections, approximate values are as follows:

- ☐ Rectangular beams = $5A/6$
- ☐ I-beams (along web direction) = Area of web
- ☐ I-beams (along flange direction) = Area of flanges
- ☐ Thin walled, hollow circular section = $A/2$
- ☐ Solid circular section = $9A/10$
- ☐ No shear deformation = $1000A$

Note

- If A_{sz} or A_{sy} equal zero, mechanisms may occur.
- For elements which support this geometric input, shear deformation effects may be removed by assigning an artificially large value.
- The section property calculator in Modeller can be used to accurately compute shear areas

Torsional Constant

The torsional constant provides a measure of the torsional rigidity of a line member. Approximate values are as follows:

Solid circle

(equivalent to the polar moment of inertia)

$$\frac{\pi \cdot r^4}{2}$$

where **r** is the radius of the circle

Hollow circle

$$\frac{\pi}{2} \left(r_2^4 - r_1^4 \right)$$

where **r2** is the outer radius

and **r1** is the inner radius

Solid square = $0.1406 a^4$

where **a** is the side length

Solid rectangle =

$$ab^3 \left[\frac{16}{3} - 3.36 \frac{b}{a} \left(1 - \frac{b^4}{12a^4} \right) \right]$$

where **2a** is the length of the longest side

and **2b** is the length of the shortest side

Equilateral triangle

$$\frac{a^4 \sqrt{3}}{80}$$

where **a** is the side length

Rectangular tube

$$\frac{2 \cdot t_1 \cdot t_2 \cdot (a - t_2)^2 (b - t_1)^2}{a t_2 + b t_1 - t_2^2 - t_1^2}$$

where

a is the length of the longest side

t1 is the thickness of the longest side

b is the length of the shortest side

t2 is the thickness of the shortest side

Thin rectangle

$$\frac{1}{3} b t^3$$

where **b** is the rectangle length

and **t** is the rectangle length thickness

Any section consisting of thin rectangles

$$\frac{1}{3} \sum b t^3$$

Solid ellipse

$$\frac{\pi a^3 b^3}{a^2 + b^2}$$

where **2a** is the longest dimension

and **2b** is the shortest dimension

Note

- The section property calculator in Modeller can be used to accurately compute torsional constants

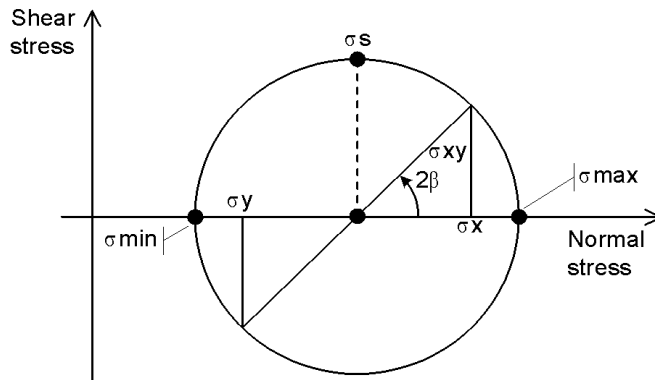
Appendix H :

Principal Stress

Output.

Output Notation for Principal Stresses

For a bi-axial stress state, the Mohr's circle representation of a stress field is:



where:

σ_{max} is the maximum principal stress.

σ_{min} is the minimum principal stress

σ_s is the maximum shear stress

β defines the orientation of the principal axis (the plane on which the principal stresses act).

$\sigma_x, \sigma_y, \sigma_{xy}$ represent an arbitrary two dimensional stress state.

Appendix I : Mass Lumping.

Mass Lumping in LUSAS

Non-Structural mass elements are used to define a lumped mass at a point, or a distributed mass along a line and over a surface.

See *Non-Structural Mass Elements* in the *Modeller Reference Manual* for more details.

Appendix J :

Moments of

Inertia.

Moments of Inertia Definitions

Second moment of area about line yy

$$I_{yy} = \int z^2 dA$$

Second moment of area about line zz

$$I_{zz} = \int y^2 dA$$

Product moment of inertia of section

$$I_{yz} = \int yz dA$$

(=0 for sections symmetric about **either** yy or zz)

First moment of area about yy

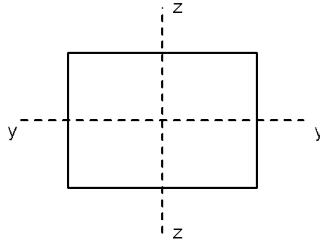
$$I_y = \int z dA$$

(=0 for sections symmetric about yy)

First moment of area about zz

$$I_z = \int y dA$$

(=0 for sections symmetric about zz)

**Note**

- The above definitions are for a section defined in the two dimensional yz plane. Similar expressions apply for a section in the three dimensional space.
- For a beam with eccentricity e from the nodal line, then:

$$I_{zz} = Ae^2 + I_{na} \text{ and } I_z = eA$$

where I_{na} is the second moment of area about the centroidal axis.

- For the purpose of the moment inertia definitions above only, the eccentricity is measured **from the nodal line to the required bending plane** (the beam's xy plane in the figure above). For example, if a beam xy plane is required such that it has negative local z coordinates relative to the nodal line, the eccentricity to be used above is negative.

Appendix K :

Results Tables.

Key to Element Results Tables

This section contains the notation for the results in the Results Tables. Some results are available in local and global directions depending on the element type. The case of the direction indicator associated for each term in the table will indicate its default direction for that element. Lower case indicates local element directions and upper case indicates that results are available in global directions by default.

Displacements

DX	Displacement in X direction	THZ	Rotation about Z
DY	Displacement in Y direction	THL1	First loof rotation
DZ	Displacement in Z direction	THL2	Second loof rotation
RSLT	Resultant displacement	DU	Hierarchical disp. at mid-node
THX	Rotation about X	DTHX	Hierarchical rotation at mid-node
THY	Rotation about Y	PRES	Pore Pressure

Note: Rotations are output in radians.

Velocities and Accelerations

VX	Velocity in X direction	AX	Acceleration in X direction
VY	Velocity in Y direction	AY	Acceleration in Y direction
VZ	Velocity in Z direction	AZ	Acceleration in Z direction
RSLT	Resultant velocity	RSLT	Resultant acceleration
VC	Results calculator values		

Strains

EX	Direct strain in X direction	Bx	Bending strain (curvature) about x axis

EY	Direct strain in Y direction	By	Bending strain (curvature) about y axis
EZ	Direct strain in Z direction	Bz	Bending strain (curvature) about z axis
EXY	Shear strain in XY plane	Bxy	Bending or torsional strain into xy plane
EYZ	Shear strain in YZ plane	Byz	Bending or torsional strain into yz plane
EZX	Shear strain in XZ plane	Bxz	Bending or torsional strain into xz plane
EMax	Maximum principal strain	BMax	Maximum principal bending strain
EMin	Minimum principal strain	BMin	Minimum principal bending strain
E1	Major principal strain	β	Angle between E1 and X axis
E2	Intermediate principal strain	EE	Equivalent strain (von Mises)
E3	Minor principal strain	EI	Maximum shear strain
Eabs	Signed largest value of principal strain		

Strains: Top/Middle/Bottom (TMB)

EX	Direct strain in X direction	E1	Major principal strain
EY	Direct strain in Y direction	E2	Intermediate principal strain
EZ	Direct strain in Z direction	E3	Minor principal strain
EXY	Shear strain in XY plane	Eabs	Signed largest value of principal strain
EYZ	Shear strain in YZ plane	β	Angle between E1 and X axis
EXZ	Shear strain in XZ plane	EE	Equivalent strain (von Mises)
		EI	Maximum shear strain

Plastic Strains

EPX	Plastic direct strain in X direction	EP1	Major principal strain
EPY	Plastic direct strain in Y direction	EP2	Intermediate principal plastic strain
EPZ	Plastic direct strain in Z direction	EP3	Minor principal plastic strain
EPXY	Plastic shear strain in XY plane	EPabs	Signed largest value of principal plastic strain
EPYZ	Plastic shear strain in YZ plane	β	Angle between EP1 and X axis

EPZX	Plastic shear strain in ZX plane	EPE	Equivalent plastic strain (von Mises)
EPMax	Maximum principal plastic strain	EPI	Maximum shear strain
EPMin	Minimum principal plastic strain	CWMax	Maximum crack width
		EFSMax	Maximum equivalent fracture strain

Creep Strains

ECX	Creep direct strain in X direction	EC1	Major principal creep strain
ECY	Creep direct strain in Y direction	EC2	Intermediate principal creep strain
ECZ	Creep direct strain in Z direction	EC3	Minor principal creep strain
ECXY	Creep shear strain in XY plane	Ecabs	Signed largest value of principal creep strain
ECYZ	Creep shear strain in YZ plane	β	Angle between EC1 and X axis
ECZX	Creep shear strain in ZX plane	ECE	Equivalent creep strain (von Mises)
ECMax	Maximum principal creep strain	ECI	Maximum shear creep strain
ECMin	Minimum principal creep strain		

Rubber Stretches

StchX	Direct stretch tensor in X direction	Stch1	Major principal stretch
StchY	Direct stretch tensor in Y direction	Stch2	Intermediate principal stretch
StchZ	Direct stretch tensor in Z direction	Stch3	Minor principal stretch
StchXY	Shear stretch tensor in XY plane	StchAbs	Signed largest value of principal stretch
StchYZ	Shear stretch tensor in YZ plane	β	Angle between Stch1 and X axis
StchXZ	Shear stretch tensor in XZ plane	StchE	Equivalent stretch
StchMax	Maximum principal stretch	StchI	Maximum shear stretch
StchMin	Minimum principal stretch		

Stresses: Continuum Elements

SX	Direct stress in global X direction	S1	Major principal stress
SY	Direct stress in global Y direction	S2	Intermediate principal stress
SZ	Direct stress in global Z direction	S3	Minor principal stress
SXY	Shear stress in xy plane	Sabs	Signed largest value of principal stress
SYZ	Shear stress in yz plane	β	Angle between E1 and x axis
SXZ	Shear stress in xz plane	SI	Maximum shear stress
SMax	Maximum principal stress	SE	Equivalent stress (von Mises)
SMin	Minimum principal stress		

Force/Moment: Bar and Beam Elements

Fx	Force in local x direction	Mx	Moment about local x direction
Fy	Force in local y direction	My	Moment about local y direction
Fz	Force in local z direction	Mz	Moment about local z direction

Stresses: Bar and Beam Elements

Sx(Fx)	Stress due to axial force in x	Sx(Fx, My)	Stress due to axial force and bending about y
Sx(My)	Stress due to bending about y	Sx(Fx, Mz)	Stress due to axial force and bending about z
Sx(Mz)	Stress due to bending about z	Sx(Fx, My, Mz)	Stress due to axial force and bending about y and z
Sx(My, Mz)	Stress due to bending about y and z		

Force/Moment: Plate Elements (per unit width)

SX	Shear force in global YZ plane	MX	Moment in global X
SY	Shear force in global XZ plane	MY	Moment in global Y
		MXY	Twisting moment in global XY plane
		Mmax	Major principal moment
		Mmin	Minor principal moment
		β	Angle between MMax and X axis

MI	Maximum shear moment
Mabs	Signed largest value of moment
ME	Equivalent moment

Force/Moment: Membrane and Shell Elements (per unit width)

Nx	In-plane force in local x direction	Mx	Moment in local x direction
Ny	In-plane force in local y direction	My	Moment in local y direction
Nxy	In-plane shear force	Mxy	Twisting moment in local xy plane
NMax	Major principal in-plane force	Mmax	Major principal moment
NMin	Minor principal in-plane force	Mmin	Minor principal moment
Nβεα	Angle between NMax and x axis	Mβεα	Angle between MMax and X axis
NI	Maximum in-plane shear force	MI	Maximum shear moment
NE	Equiv stress resultant (von Mises)	ME	Equivalent moment
Nabs	Signed largest value of in-plane force	Mabs	Signed largest value of moment
Sx	Shear force in local yz plane		
Sy	Shear force in local xz plane		

Stresses: Top/Middle/Bottom (TMB)

SX	Direct stress in global X direction	S1	Major principal stress
SY	Direct stress in global Y direction	S2	Intermediate principal stress
SZ	Direct stress in global Z direction	S3	Minor principal stress
SXY	Shear stress in XY plane	Sabs	Signed largest value of principal stress
SYZ	Shear stress in YZ plane	SI	Maximum shear stress
SXZ	Shear stress in XZ plane	SE	Equivalent stress (von Mises)

Force/Moment: Wood-Armer (per unit width for Shells)

Mx(T)	Top surface local x moment	Nx(T)	Top surface local x force
My(T)	Top surface local y moment	Ny(T)	Top surface local y force
Mx(B)	Bottom surface local x moment	Nx(B)	Bottom surface local x force
My(B)	Bottom surface local y moment	Ny(B)	Bottom surface local y force
Util(T)	Top surface utilisation factor	Fc(T)	Top surface concrete force
Util(B)	Bottom surface utilisation factor	Fc(B)	Bottom surface concrete force
MUtil(T)	Top surface utilisation factor for bending only		
MUtil(B)	Bottom surface utilisation factor for bending only		

Force/Moment: Wood-Armer (per unit width for Plates and Grillages)

MX(T)	Top surface global X moment	MUtil(T)	Top surface utilisation factor for bending only
MY(T)	Top surface global y moment	MUtil(B)	Bottom surface utilisation factor for bending only
MX(B)	Bottom surface global x moment		
MY(B)	Bottom surface global y moment		

Additional Force/Moment Components

Note for influence analysis when a reference path has been specified, additional force/moment components are available for selection when transforming results. These are not listed for relevant elements in the Results tables.

FV (longitudinal)	Force in Vertical direction for longitudinal members that are following the reference path	MF (longitudinal)	Flexural Moment in longitudinal members that are following the reference path
FV (transverse)	Force in Vertical direction for transverse members that are orthogonal or skewed in relation to the reference path	MF (transverse)	Flexural Moment in transverse members that are orthogonal or skewed in relation to the reference path

Stresses: Interface Elements

Sx	Shear traction in local x direction	Sy	Shear traction in local y direction
Sz	Direct traction in the thickness direction		

Potential

PHI	Field variable	PHIC	Results calculator values

Gradients

GX	Field gradient in X direction
GY	Field gradient in Y direction
GZ	Field gradient in Z direction

Fluxes

qX	Field flux in X direction
qY	Field flux in Y direction
qZ	Field flux in Z direction

Reactions / Residual Forces

FX	Force in X direction	MZ	Moment about Z axis
FY	Force in Y direction	FDU	Force due to hierarchical displacement
FZ	Force in Z direction	MDX	Moment due to hierarchical rotation
RSLT	Resultant force		
MX	Moment about X axis	QC	Flow at a point (field problems)
MY	Moment about Y axis	VFL	Velocity of Flow
		W	

Reaction Stress

PX	Stress due to reaction in X direction	PZ	Stress due to reaction in Z direction
PY	Stress due to reaction in Y direction		

Fatigue Parameters

Damage	A measure of damage	LogLife	Log repeats to failure

Note. The fatigue facility uses miner's rule, that is:

$$n1/N1 + n2/N2 + \dots + ni/Ni = \text{Damage}$$

where *Damage* is the damage variable and is usually taken as unity (experiment usually gives values between 0.7 and 2.2). *ni* is the number of cycles of stress applied to the structure and *Ni* is the life corresponding to the

stress. `Loglife` is the log (base 10) of the life expectancy of the structure according to the loading and the number of cycles specified. Life is measured in terms of cycles.

Damage Parameters

DDAMA Damage variable **DAMAM** Damage consistency parameter
CCURD Damage threshold **DFUNC** Damage function

***Note.** Damage parameters are only available when a damage model is in use.*

Strain Energy and Plastic Work

SED Strain energy density **PWD** Plastic work density
(`StEngD`)

***Note.** Strain energy density and plastic work density values can be accessed if turned on by selecting **Calculate Strain Energy and Plastic Work Densities** from the **Results > Options** dialog or by using the command: **SET RESULTS ENERGY**.*

Adaptive Error

Eadp Adaptive error.

***Note.** Adaptive error results are only available when an adaptive results column is set. See the *LUSAS User Manual* for more details.*

State Variables

State variables can be accessed with the command:

SET RESULTS STATE_VARIABLES istvb nsvcmp isvloc

Where `istvb` is the type of state variable required, `nsvcmp` is the number of state variables required, and `isvloc` is the start location of the first state variable required.

The results columns for these state variables vary according to the results type set. The column descriptors have the following prefixes:

- ☐ **PL** Plastic, Rubber
- ☐ **CR** Creep
- ☐ **DM** Damage

- followed by the number of the state variable required. For example, if four creep state variables are required, the column descriptors will be CR1, CR2, CR3 and CR4.

Transforming Results Directions

Important: Some results entities can be transformed. The results components will use alternative suffixes if results are calculated relative to a system other than the global axis set. The element results tables show the default results directions for all elements with lower case subscripts being used for local results.

See the [Local and Global Results](#) in the *LUSAS Modeller User Manual* for details of results transformation procedures.

2D Structural Bars BAR2, BAR3

Entity			Component								
Displacement	DX	DY	RSLT								
Force/Moment	FX	Fabs	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp
Strain	EX	Eabs	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp		
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp				
Loading	FX	FY	RSLT								
Reaction	FX	FY	RSLT								
Residual Force	FX	FY	RSLT								
Reaction Stress											
Velocity	VX	VY	RSLT								
Acceleration	AX	AY	RSLT								
Plastic Strain	EPX	EPabs	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp		
Creep Strain	ECX	ECabs	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp		
Rubber Stretches											
TMB Stress											
TMB Strain											
TMB Plastic Strain											
TMB Creep Strain											

3D Structural Bars **BRS2**, **BRS3**

Entity		Component									
Displacement	DX	DY	DZ	RSLT							
Force/Moment	FX	Fabs	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp
Strain	EX	Eabs	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp		
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp				
Loading	FX	FY	FZ	RSLT							
Reaction	FX	FY	FZ	RSLT							
Residual Force	FX	FY	FZ	RSLT							
Reaction Stress											
Velocity	VX	VY	VZ	RSLT							
Acceleration	AX	AY	AZ	RSLT							
Plastic Strain	EPX	EPabs	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp		
Creep Strain	ECX	ECabs	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp		
Rubber Stretches											
TMB Stress											
TMB Strain											
TMB Plastic Strain											
TMB Creep Strain											

2D Engineering Beam **BEAM**

Entity				Component			
Displacement	DX	DY	RSLT	THZ			
Force/Moment	Fx	Fy	Mz	Damage	LogLife	SED	Eadp
Strain							
Loading	FX	FY	RSLT	MZ			
Reaction	FX	FY	RSLT	MZ			
Residual Force							
Reaction Stress							
Velocity	VX	VY	RSLT				
Acceleration	AX	AY	RSLT				
Plastic Strain							
Creep Strain							
Rubber Stretches							
TMB Stress							
TMB Strain							
TMB Plastic Strain							
TMB Creep Strain							

3D Engineering Thick Beam **BMS3**

Entity					Component						
Displacement	DX	DY	DZ	RSLT	THX	THY	THZ				
Force/Moment	Fx	Fy	Fz	Mx	My	Mz	Damage	LogLife	SED	Eadp	
Strain											
Loading	FX	FY	FZ	RSLT	MX	MY	MZ				
Reaction	FX	FY	FZ	RSLT	MX	MY	MZ				
Residual Force											
Reaction Stress											
Velocity	VX	VY	VZ	RSLT							
Acceleration	AX	AY	AZ	RSLT							
Plastic Strain											
Creep Strain											
Rubber Stretches											
TMB Stress											
TMB Strain											
TMB Plastic Strain											
TMB Creep Strain											

2D Engineering Grillage Thick Beam **GRIL**

Entity	Component												
Displacement	DZ	RSLT	THX	THY									
Force/Moment	Fz	Mx	My	Mx(T)	My(T)	Mx(B)	My(B)	Util(T)	Util(B)	Damage	LogLife	SED	Eadp
Strain													
Loading	FZ	RSLT	MX	MY									
Reaction	FZ	RSLT	MX	MY									
Residual Force													
Reaction Stress													
Velocity	VZ	RSLT											
Acceleration	AZ	RSLT											
Plastic Strain													
Creep Strain													
Rubber Stretches													
TMB Stress													
TMB Strain													
TMB Plastic Strain													
TMB Creep Strain													

Note: Wood-Armer results are only available for plotting /printing at nodes. They are not available unaveraged at nodes within elements or at Gauss points.

3D Thick Beam (Nonlinear) [BTS3](#)

Entity					Component							
Displacement	DX	DY	DZ	RSLT	THX	THY	THZ					
Force/Moment	Fx	Fy	Fz	Mx	My	Mz	Damage	LogLife	SED	PWD	Eadp	
Strain	Ex	Ey	Ez	Bx	By	Bz	SED	PWD	Eadp			
Loading	FX	FY	FZ	RSLT	MX	MY	MZ					
Reaction	FX	FY	FZ	RSLT	MX	MY	MZ					
Residual Force	FX	FY	FZ	RSLT	MX	MY	MZ					
Reaction Stress												
Velocity	VX	VY	VZ	RSLT								
Acceleration	AX	AY	AZ	RSLT								
Plastic Strain												
Creep Strain												
Rubber Stretches												
TMB Stress												
TMB Strain												
TMB Plastic Strain												
TMB Creep Strain												

3D Thick Beam Elements [BMI21](#), [BMI22](#), [BMI31](#), [BMI33](#), [BMX21](#), [BMX22](#), [BMX31](#), [BMX33](#)

Entity				Component											
Displacement	DX	DY	DZ	RSLT	THX	THY	THZ								
Force.Moment	Fx	My	Mz	Mx	My	Mz	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp
Strain	Ex	By	Bz	Bx	By	Bz	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp		
Loading	FX	FY	FZ	RSLT	MX	MY	MZ								
Reaction	FX	FY	FZ	RSLT	MX	MY	MZ								
Residual Force	FX	FY	FZ	RSLT	MX	MY	MZ								
Reaction Stress															
Velocity	VX	VY	VZ	RSLT											
Acceleration	AX	AY	AZ	RSLT											
Plastic Strain	EPx	EPxy	EPzx	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp					
Creep Strain	ECx	ECxy	ECzx	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp					
Rubber Stretches															
TMB Stress															
TMB Strain															
TMB Plastic Strain															
TMB Creep Strain															

Note: Plastic and creep strains are only available for *BMX21*, *BMX31*, *BMX22*, *BMX33* elements with the appropriate material models.

2D Kirchhoff Thin Beams **BM3**, **BMX3**

Entity			Component									
Displacement	DX	DY	RSLT	THZ	DU							
Force/Moment	Fx	Fy	Mz	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp
Strain	Ex	Ey	Bz	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp		
Loading	FX	FY	RSLT	MZ	FDU							
Reaction	FX	FY	RSLT	MZ	FDU							
Residual Force	FX	FY	RSLT	MZ	FDU							
Reaction Stress												
Velocity	VX	VY	RSLT									
Acceleration	AX	AY	RSLT									
Plastic Strain	EPx	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp				
Creep Strain	ECx	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp				
Rubber Stretches												
TMB Stress												
TMB Strain												
TMB Plastic Strain												
TMB Creep Strain												

Note: Plastic and creep strains are only available for BMX3 elements with the appropriate material models.

3D Kirchhoff Thin Beams **BS3**, **BS4**, **BSX4**

Entity		Component									
Displacement	DX	DY	DZ	RSLT	THX	THY	THZ	DU	DTHX		
Force/Moment	Fx	My	Mz	Tzx	Txy	Fy	Fz	Damage	LogLife	DDAMA	CURRD
(continued)	Eadp										
Strain	Ex	By	Bz	Bzx	Bxy	Ey	Ez				
(continued)	Eadp										
Loading	FX	FY	FZ	RSLT	MX	MY	MZ	FDU	MDX		
Reaction	FX	FY	FZ	RSLT	MX	MY	MZ	FDU	MDX		
Residual Force	FX	FY	FZ	RSLT	MX	MY	MZ	FDU	MDX		
Reaction Stress											
Velocity	VX	VY	VZ	RSLT							
Acceleration	AX	AY	AZ	RSLT							
Plastic Strain	EPx	EPxy	EPzx	EPyz	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp
Creep Strain	ECx	ECxy	ECzx	ECyz	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp
Rubber Stretches											
TMB Stress											
TMB Strain											
TMB Plastic Strain											
TMB Creep Strain											

Note: Plastic and creep strains are only available for BSX4 elements with the appropriate material models.

3D Semiloof Thin Beams [BSL3](#), [BSL4](#), [BXL4](#)

Entity					Component										
Displacement	DX	DY	DZ	RSLT	THX	THY	THZ	THL1	THL2						
Force.Moment	Fx	My	Mz	Tzx	Txy	Fy	Fz	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD
(continued)	Eadp														
Strain	Ex	By	Bz	Bzx	Bxy	Ey	Ez	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp	
Loading	FX	FY	FZ	RSLT	MX	MY	MZ	ML1	ML2						
Reaction	FX	FY	FZ	RSLT	MX	MY	MZ	ML1	ML2						
Residual Force	FX	FY	FZ	RSLT	MX	MY	MZ	ML1	ML2						
Reaction Stress															
Velocity	VX	VY	VZ	RSLT											
Acceleration	AX	AY	AZ	RSLT											
Plastic Strain	EPx	EPxy	EPyz	EPzx	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp				
Creep Strain	ECx	ECxy	ECyz	ECzx	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp				
Rubber Stretches															
TMB Stress															
TMB Strain															
TMB Plastic Strain															
TMB Creep Strain															

Note: Plastic and creep strains are only available for BXL4 elements with the appropriate material models.

2D Continuum (Plane Stress) TPM3/6, QPM4/8, QPM4M, TPK6, QPK8

Entity		Component									
Displacement	DX	DY	RSLT								
Stress	SX	SY	SXY	SMax	SMin	SI	β	Sabs	SE		
(continued)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp		
Strain	EX	EY	EXY	EMax	EMin	EI	β	Eabs	EE		
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp				
Loading	FX	FY	RSLT								
Reaction	FX	FY	RSLT								
Residual Force	FX	FY	RSLT								
Reaction Stress	PX	PY									
Velocity	VX	VY	RSLT								
Acceleration	AX	AY	RSLT								
Plastic Strain	EPX	EPY	EPXY	EPMax	EPMin	EPI	β	EPabs	EPE	CWMax	EFSMax
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp				
Creep Strain	ECX	ECY	ECXY	ECMax	ECMin	ECI	β	ECabs	ECE		
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp				
Rubber Stretches	StchX	StchY	StchXY	StchMax	StchMin	Stchl	β	StchAbs	StchE		
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp				
TMB Stress											
TMB Strain											
TMB Plastic Strain											
TMB Creep Strain											

Notes:

Rubber stretches are only available for QPM4M elements with rubber material models. Strains are not available for this element when using rubber materials.

Plastic strain components CWMax and EFSMax are only available when the Smoothed Multi-crack Concrete Model (Model 102) is used.

2D Continuum Plane Stress (Explicit Dynamics)

TPM3E, QPM4E

Entity		Component							
Displacement	DX	DY	RSLT						
Stress	SX	SY	SXY	SMax	SMin	SI	β	Sabs	SE
(continued)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp
Strain	EX	EY	EXY	EMax	EMin	EI	β	Eabs	EE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp		
Loading	FX	FY	RSLT						
Reaction	FX	FY	RSLT						
Residual Force	FX	FY	RSLT						
Reaction Stress	PX	PY							
Velocity	VX	VY	RSLT						
Acceleration	AX	AY	RSLT						
Plastic Strain	EPX	EPY	EPXY	EPMax	EPMin	EPI	β	EPabs	EPE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp		
Creep Strain	ECX	ECY	ECXY	ECMax	ECMin	ECl	β	ECabs	ECE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp		
Rubber Stretches									
TMB Stress									
TMB Strain									
TMB Plastic Strain									
TMB Creep Strain									

2D Continuum (Plane Strain) TPN3/6, QPN4/8, TNK6, QNK8, QPN4M

Entity	Component											
Displacement	DX	DY	RSLT									
Stress	SX	SY	SXY	SZ	S1	S2	S3	SI	Sabs	SE		
(continued)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp			
Strain	EX	EY	EXY	EZ	E1	E2	E3	EI	Eabs	EE		
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp					
Loading	FX	FY	RSLT									
Reaction	FX	FY	RSLT									
Residual Force	FX	FY	RSLT									
Reaction Stress	PX	PY										
Velocity	VX	VY	RSLT									
Acceleration	AX	AY	RSLT									
Plastic Strain	EPX	EPY	EPXY	EPZ	EP1	EP2	EP3	EPI	EPabs	EPE	CWMax	EFSMax
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp					
Creep Strain	ECX	ECY	ECXY	ECZ	EC1	EP2	EC3	ECI	ECabs	ECE		
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp					
Rubber Stretches	StchX	StchY	StchXY	StchZ	Stch1	Stch2	Stch3	Stchl	StchAbs	StchE		
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp					
TMB Stress												
TMB Strain												
TMB Plastic Strain												
TMB Creep Strain												

Notes:

Rubber stretches are only available for QPN4M elements with rubber material models. Strains are not available for this element when using rubber materials.

Plastic strain components CWMax and EFSMax are only available when the Smoothed Multi-crack Concrete Model (Model 102) is used.

2D Continuum (Plane Strain) [QPN4L](#)

Entity		Component							
Displacement	DX	DY	RSLT						
Stress	SX	SY	SXY	SZ	S1	S2	S3	SI	SE
Strain	StchX	StchY	StchXY	StchZ	Stch1	Stch2	Stch3	Stchl	StchE
Loading	FX	FY	RSLT						
Reaction	FX	FY	RSLT						
Residual Force	FX	FY	RSLT						
Reaction Stress	PX	PY							
Velocity	VX	VY	RSLT						
Acceleration	AX	AY	RSLT						
Plastic Strain	EPX	EPY	EPXY	EPZ	EP1	EP2	EP3	EPI	EPE
Creep Strain									
Rubber Stretches	StchX	StchY	StchXY	StchZ	Stch1	Stch2	Stch3	Stchl	StchE
TMB Stress									
TMB Strain									
TMB Plastic Strain									
TMB Creep Strain									

2D Plain Strain Two Phase Continuum **TPN6P**, **QPN8P**

Entity	Component									
Displacement	DX	DY	RSLT	Pres						
Stress	SX	SY	SXY	SZ	S1	S2	S3	SI	Sabs	SE
(continued)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp	
Strain	EX	EY	EXY	EZ	E1	E2	E3	EI	Eabs	EE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp			
Loading	FX	FY	RSLT	Q						
Reaction	FX	FY	RSLT	Q						
Residual Force	FX	FY	RSLT							
Reaction Stress	PX	PY								
Velocity	VX	VY	RSLT							
Acceleration	AX	AY	RSLT							
Plastic Strain	EPX	EPY	EPXY	EPZ	EP1	EP2	EP3	EPI	EPabs	EPE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp	CWMax	EFSTMax	
Creep Strain	ECX	ECY	ECXY	ECZ	EC1	EP2	EC3	ECI	ECabs	ECE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp			
Rubber Stretches										
TMB Stress										
TMB Strain										
TMB Plastic Strain										
TMB Creep Strain										

Notes:

Rubber stretches are only available for QPN4M elements with rubber material models. Strains are not available for this element when using rubber materials.

Plastic strain components CWMax and EFSTMax are only available when the Smoothed Multi-crack Concrete Model (Model 102) is used.

2D Continuum Plane Strain (Explicit Dynamics) [TPN3E](#), [QPN4E](#)

Entity			Component							
Displacement	DX	DY	RSLT							
Stress	SX	SY	SXY	SZ	S1	S2	S3	SI	Sabs	SE
(continued)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp	
Strain	EX	EY	EXY	EZ	E1	E2	E3	EI	Eabs	EE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp			
Loading	FX	FY	RSLT							
Reaction	FX	FY	RSLT							
Residual Force	FX	FY	RSLT							
Reaction Stress	PX	PY								
Velocity	VX	VY	RSLT							
Acceleration	AX	AY	RSLT							
Plastic Strain	EPX	EPY	EPXY	EPZ	EP1	EP2	EP3	EPI	EPabs	EPE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp			
Creep Strain	ECX	ECY	ECXY	ECZ	EC1	EP2	EC3	ECI	ECabs	ECE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp			
Rubber Stretches										
TMB Stress										
TMB Strain										
TMB Plastic Strain										
TMB Creep Strain										

2D Continuum Axisymmetric Solid (Explicit Dynamics)

TAX3E, QAX4E

Entity		Component								
Displacement	DX	DY	RSLT	Pres						
Stress	SX	SY	SXY	SZ	S1	S2	S3	SI	Sabs	SE
(continued)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp	
Strain	EX	EY	EXY	EZ	E1	E2	E3	EI	Eabs	EE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp			
Loading	FX	FY	RSLT							
Reaction	FX	FY	RSLT							
Residual Force	FX	FY	RSLT							
Reaction Stress	PX	PY								
Velocity	VX	VY	RSLT							
Acceleration	AX	AY	RSLT							
Plastic Strain	EPX	EPY	EPXY	EPZ	EP1	EP2	EP3	EPI	EPabs	EPE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp			
Creep Strain	ECX	ECY	ECXY	ECZ	EC1	EP2	EC3	ECI	ECabs	ECE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp			
Rubber Stretches										
TMB Stress										
TMB Strain										
TMB Plastic Strain										
TMB Creep Strain										

2D Axisymmetric Solid Two Phase Continuum **TAX6P**, **QAX8P**

Entity		Component									
Displacement	DX	DY	RSLT	Pres							
Stress	SX	SY	SXY	SZ	S1	S2	S3	SI	Sabs	SE	
(continued)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp		
Strain	EX	EY	EXY	EZ	E1	E2	E3	EI	Eabs	EE	
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp				
Loading	FX	FY	RSLT	Q							
Reaction	FX	FY	RSLT	Q							
Residual Force	FX	FY	RSLT								
Reaction Stress	PX	PY									
Velocity	VX	VY	RSLT								
Acceleration	AX	AY	RSLT								
Plastic Strain	EPX	EPY	EPXY	EPZ	EP1	EP2	EP3	EPI	EPabs	EPE	
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp	CWMax	EFSMax		
Creep Strain	ECX	ECY	ECXY	ECZ	EC1	EP2	EC3	ECI	ECabs	ECE	
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp				
Rubber Stretches											
TMB Stress											
TMB Strain											
TMB Plastic Strain											
TMB Creep Strain											

Notes:

Plastic strain components CWMax and EFSMax are only available when the Smoothed Multi-crack Concrete Model (Model 102) is used.

2D Continuum Axisymmetric Solid Fourier **TAX3/6F**, **QAX4/8F**

Entity		Component								
Displacement	DX	DY	DZ	RSLT						
Stress	SX	SY	SXY	SZ	S1	S2	S3	SI	Sabs	SE
(continued)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	Eadp		
Strain	EX	EY	EXY	EZ	E1	E2	E3	EI	Eabs	EE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	Eadp				
Loading	FX	FY	FZ	RSLT						
Reaction	FX	FY	FZ	RSLT						
Residual Force										
Reaction Stress	PX	PY								
Velocity	VX	VY	VZ	RSLT						
Acceleration	AX	AY	AZ	RSLT						
Plastic Strain										
Creep Strain										
Rubber Stretches										
TMB Stress										
TMB Strain										
TMB Plastic Strain										
TMB Creep Strain										

Axisymmetric Solid **TAX3/6**, **QAX4/8**, **QAX4M**, **TXK6**, **QXK8**

Entity			Component									
Displacement	DX	DY	RSLT									
Stress	SX	SY	SXY	SZ	S1	S2	S3	SI	Sabs	SE		
(continued)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp			
Strain	EX	EY	EXY	EZ	E1	E2	E3	EI	Eabs	EE		
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp					
Loading	FX	FY	RSLT									
Reaction	FX	FY	RSLT									
Residual Force	FX	FY	RSLT									
Reaction Stress	PX	PY										
Velocity	VX	VY	RSLT									
Acceleration	AX	AY	RSLT									
Plastic Strain	EPX	EPY	EPXY	EPZ	EP1	EP2	EP3	EPI	EPabs	EPE	CWMax	EFSMax
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp					
Creep Strain	ECX	ECY	ECXY	ECZ	EC1	EP2	EC3	ECI	ECabs	ECE		
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp					
Rubber Stretches	StchX	StchY	StchXY	StchZ	Stch1	Stch2	Stch3	Stchl	StchAbs	StchE		
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp					
TMB Stress												
TMB Strain												
TMB Plastic Strain												
TMB Creep Strain												

Notes:

Rubber stretches are only available for QAX4M elements with rubber material models. Strains are not available for this element when using rubber materials.

Plastic strain components CWMax and EFSMax are only available when the Smoothed Multi-crack Concrete Model (Model 102) is used.

Axisymmetric Solid Large Strain **QAX4L**

Entity		Component								
Displacement	DX	DY	RSLT	Pres						
Stress	SX	SY	SXY	SZ	S1	S2	S3	SI	Sabs	SE
(continued)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp	
Strain	StchX	StchY	StchXY	StchZ	Stch1	Stch2	Stch3	StchI	StchE	
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp			
Loading	FX	FY	RSLT							
Reaction	FX	FY	RSLT							
Residual Force	FX	FY	RSLT							
Reaction Stress	PX	PY								
Velocity	VX	VY	RSLT							
Acceleration	AX	AY	RSLT							
Plastic Strain	EPX	EPY	EPXY	EPZ	EP1	EP2	EP3	EPI	EPE	
Creep Strain										
Rubber Stretches	StchX	StchY	StchXY	StchZ	Stch1	Stch2	Stch3	StchI	StchE	
TMB Stress										
TMB Strain										
TMB Plastic Strain										
TMB Creep Strain										

3D Solid Continuum [TH4/10](#), [TH10S](#), [PN6/12/15](#), [PN6L/12L](#), [HX8/16/20](#), [HX8M](#), [HX8L/16L](#), [TH10K](#), [PN15K](#), [HX20K](#)

Entity	Component											
Displacement	DX	DY	DZ	RSLT								
Stress	SX	SY	SZ	SXY	SYZ	SZX	S1	S2	S3	SI	Sabs	SE
(continued)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp			
Strain	EX	EY	EZ	EXY	EYZ	EZX	E1	E2	E3	EI	Eabs	EE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp					
Loading	FX	FY	FZ	RSLT								
Reaction	FX	FY	FZ	RSLT								
Residual Force	FX	FY	FZ	RSLT								
Reaction Stress	PX	PY	PZ									
Velocity	VX	VY	VZ	RSLT								
Acceleration	AX	AY	AZ	RSLT								
Plastic Strain	EPX	EPY	EPZ	EPXY	EPYZ	EPZX	EP1	EP2	EP3	EPI	EPabs	EPE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp	CWMax	EFSMax			
Creep Strain	ECX	ECY	ECZ	ECXY	ECYZ	ECZX	EC1	EC2	EC3	ECI	ECabs	ECE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp					
Rubber Stretches	StchX	StchY	StchZ	StchXY	StchYZ	StchZX	Stch1	Stch2	Stch3	Stchl	StchAbs	StchE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp					
TMB Stress												
TMB Strain												
TMB Plastic Strain												
TMB Creep Strain												

Notes:

Rubber stretches are only available for HX8M elements with rubber material models.

Strains are not available for this element when using rubber materials.

Plastic strain components CWMax and EFSMax are only available when the Smoothed Multi-crack Concrete Model (Model 102) is used.

3D Solid Continuum Two Phase TH10P, PN12P, PN15P, HX16P, HX20P

Entity		Component										
Displacement	DX	DY	DZ	RSLT	Pres							
Stress	SX	SY	SZ	SXY	SYZ	SZX	S1	S2	S3	SI	Sabs	SE
(continued)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp			
Strain	EX	EY	EZ	EXY	EYZ	EZX	E1	E2	E3	EI	Eabs	EE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp					
Loading	FX	FY	FZ	RSLT	Q							
Reaction	FX	FY	FZ	RSLT	Q							
Residual Force	FX	FY	FZ	RSLT								
Reaction Stress	PX	PY	PZ									
Velocity	VX	VY	VZ	RSLT								
Acceleration	AX	AY	AZ	RSLT								
Plastic Strain	EPX	EPY	EPZ	EPXY	EPYZ	EPZX	EP1	EP2	EP3	EPI	EPabs	EPE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp	CWMax	EFSTMax			
Creep Strain	ECX	ECY	ECZ	ECXY	ECYZ	ECZX	EC1	EC2	EC3	ECI	ECabs	ECE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp					
Rubber Stretches												
TMB Stress												
TMB Strain												
TMB Plastic Strain												
TMB Creep Strain												

Notes:

Plastic strain components *CWMax* and *EFSTMax* are only available when the Smoothed Multi-crack Concrete Model (Model 102) is used.

3D Solid Continuum Explicit Dynamics [TH4E](#), [PN6E](#), [HX8E](#)

Entity				Component									
Displacement	DX	DY	DZ	RSLT	Pres								
Stress	SX	SY	SZ	SXY	SYZ	SZX	S1	S2	S3	SI	Sabs	SE	
(continued)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp				
Strain													
Loading	FX	FY	FZ	RSLT									
Reaction	FX	FY	FZ	RSLT									
Residual Force	FX	FY	FZ	RSLT									
Reaction Stress	PX	PY	PZ										
Velocity	VX	VY	VZ	RSLT									
Acceleration	AX	AY	AZ	RSLT									
Plastic Strain													
Creep Strain													
Rubber Stretches													
TMB Stress													
TMB Strain													
TMB Plastic Strain													
TMB Creep Strain													

Isoflex Thin Plates **TF3**, **QF4**

Entity		Component													
Displacement	DZ	RSLT	THX	THY											
Stress	MX	MY	MX _Y	MM _{ax}	MM _{in}	MI	β	Nabs	ME	Mx(T)	My(T)	Mx(B)	My(B)	Util(T)	Util(B)
(continued)	Damage	LogLife	SED	PWD	Eadp										
Strain	BX	BY	BXY	BMax	BMin	BI	β	Eabs	BE	SED	PWD	Eadp			
Loading	FZ	RSLT	MX	MY											
Reaction	FZ	RSLT	MX	MY											
Residual Force	FZ	RSLT	MX	MY											
Reaction Stress	PZ														
Velocity	VZ	RSLT													
Acceleration	AZ	RSLT													
Plastic Strain															
Creep Strain															
Rubber Stretches															
TMB Stress	SX	SY	SXY	SMax	SMin	SI	β	Sabs	SE	Damage	LogLife	SED	PWD	Eadp	
TMB Strain	EX	EY	EXY	EMax	EMin	EI	β	Eabs	EE	SED	PWD	Eadp			
TMB Plastic Strain															
TMB Creep Strain															

Isoflex Thick Plates **QSC4**

Entity	Component															
Displacement	DZ	RSLT	THX	THY												
Stress	MX	MY	MXY	Sx	Sy	MMax	MMin	MI	β	Nabs	ME	Mx(T)	My(T)	Mx(B)	My(B)	
(continued)	Util(T)	Util(B)	Damage	LogLife	SED	PWD	Eadp									
Strain	BX	BY	BXY	EZX	EYZ	BMax	BMin	BI	β	Eabs	BE	SED	PWD	Eadp		
Loading	FZ	RSLT	MX	MY												
Reaction	FZ	RSLT	MX	MY												
Residual Force	FZ	RSLT	MX	MY												
Reaction Stress	PZ															
Velocity	VZ	RSLT														
Acceleration	AZ	RSLT														
Plastic Strain																
Creep Strain																
Rubber Stretches																
TMB Stress	SX	SY	SXY	SMax	SMin	SI	β	Sabs	SE	Damage	LogLife	SED	PWD	Eadp		
TMB Strain	EX	EY	EXY	EMax	EMin	EI	β	Eabs	EE	SED	PWD	Eadp				
TMB Plastic Strain																
TMB Creep Strain																

Mindlin Thick Plates TTF6, QTF8

Entity		Component														
Displacement	DZ	RSLT	THX	THY												
Stress	MX	MY	MXY	Sx	Sy	MMax	MMin	MI	β	Nabs	ME	Mx(T)	My(T)	Mx(B)	My(B)	
(continued)	Util(T)	Util(B)	Damage	LogLife	SED	PWD	Eadp									
Strain	BX	BY	BXY	EZX	EYZ	BMax	BMin	BI	β	Eabs	BE	SED	PWD	Eadp		
Loading	FZ	RSLT	MX	MY												
Reaction	FZ	RSLT	MX	MY												
Residual Force	FZ	RSLT	MX	MY												
Reaction Stress	PZ															
Velocity	VZ	RSLT														
Acceleration	AZ	RSLT														
Plastic Strain																
Creep Strain																
Rubber Stretches																
TMB Stress	SX	SY	SXY	SMax	SMin	SI	β	Sabs	SE	Damage	LogLife	SED	PWD	Eadp		
TMB Strain	EX	EY	EXY	EMax	EMin	EI	β	Eabs	EE	SED	PWD	Eadp				
TMB Plastic Strain																
TMB Creep Strain																

2D Axisymmetric Membranes **BXM2**, **BXM3**

Entity	Component								
Displacement	DX	DY	RSLT						
Stress	Sx	Sz	SMax	SMin	SI	β	Sabs	SE	
(continued)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp
Strain	Ex	Ez	EMax	EMin	EI	β	Eabs	EE	
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp		
Loading	FX	FY	RSLT						
Reaction	FX	FY	RSLT						
Residual Force	FX	FY	RSLT						
Reaction Stress	PX	PY							
Velocity	VX	VY	RSLT						
Acceleration	AX	AY	RSLT						
Plastic Strain	EPx	EPz	EPMax	EPMin	EPI	β	EPabs	EPE	
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp		
Creep Strain	ECx	ECz	ECMax	ECMin	ECI	β	ECabs	ECE	
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp		
Rubber Stretches									
TMB Stress									
TMB Strain									
TMB Plastic Strain									
TMB Creep Strain									

Note:

Rubber models are available for use with the BXM2 element, however strains are output and rubber stretches are not available.

3D Space Membranes TSM3, SMI4

Entity				Component								
Displacement	DX	DY	DZ	RSLT								
Stress	Nx	Ny	Nxy	NMax	NMin	Ns	β	Nabs	Ne			
(continued)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp			
Strain	Ex	Ey	Exy	EMax	EMin	EI	β	Eabs	EE			
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp					
Loading	FX	FY	FZ	RSLT								
Reaction	FX	FY	FZ	RSLT								
Residual Force	FX	FY	FZ	RSLT								
Reaction Stress	PX	PY	PZ									
Velocity	VX	VY	VZ	RSLT								
Acceleration	AX	AY	AZ	RSLT								
Plastic Strain												
Creep Strain												
Rubber Stretches												
TMB Stress	SX	SY	SZ	SXY	SYZ	SZX	S1	S2	S3	SI	Sabs	SE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp					
TMB Strain	EX	EY	EZ	EXY	EYZ	EZX	E1	E2	E3	EI	Eabs	EE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp					
TMB Plastic Strain												
TMB Creep Strain												

Axisymmetric Shells **BXS3**

Entity		Component									
Displacement	DX	DY	RSLT	THZ	DU						
Stress	Nx	Nz	Mx	Mz	Ny	NMax	NMin	Ns	β	Nabs	Ne
(continued)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp		
Strain	Ex	Ez	Bx	Bz	Ey	EMax	EMin	EI	β	Eabs	EE
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp				
Loading	FX	FY	RSLT	MZ	FDU						
Reaction	FX	FY	RSLT	MZ	FDU						
Residual Force	FX	FY	RSLT	MZ	FDU						
Reaction Stress	PX	PY									
Velocity	VX	VY	RSLT								
Acceleration	AX	AY									
Plastic Strain											
Creep Strain											
Rubber Stretches											
TMB Stress	Sx	Sz	SMax	SMin	SI	β	Sabs	SE			
(continued)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp		
TMB Strain	Ex	Ez	EPMx	EMin	EI	β	Eabs	EE			
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp				
TMB Plastic Strain	EPx	EPz	EPMx	EPMin	EPI	β	EPabs	EPE			
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp				
TMB Creep Strain	ECx	ECz	ECMax	ECMin	ECI	β	ECabs	ECE			
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp				

3D Flat Thin Shells **TS3**, **QSI4**

Entity				Component											
Displacement	DX	DY	DZ	RSLT	THX	THY	THZ								
Stress	Nx	Ny	Nxy	Mx	My	Mxy	NMax	NMin	Ns	β	Nabs	Ne	Nx(T)/ Mx(T)	Ny(T)/ Ny(T)	Nx(B)/ Mx(B)
(continued)	Ny(B)/ My(B)	Util(T)	Util(B)	MUtil(T)	MUtil(B)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	Fc(T)	Fc(B)	Eadp
Strain	Ex	Ey	Exy	Bx	By	Bxy	EMax	EMin	EI	β	Eabs	EE			
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	Eadp									
Loading	FX	FY	FZ	RSLT	MX	MY	MZ								
Reaction	FX	FY	FZ	RSLT	MX	MY	MZ								
Residual Force	FX	FY	FZ	RSLT	MX	MY	MZ								
Reaction Stress	PX	PY	PZ												
Velocity	VX	VY	VZ	RSLT											
Acceleration	AX	AY	AZ	RSLT											
Plastic Strain															
Creep Strain															
Rubber Stretches															
TMB Stress	SX	SY	SZ	SXY	SYZ	SZX	S1	S2	S3	SI	Sabs	SE			
(continued)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	Eadp							
TMB Strain	EX	EY	EZ	EXY	EYZ	EZX	E1	E2	E3	EI	Eabs	EE			
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp								
TMB Plastic Strain															
TMB Creep Strain															

3D Flat Thin Nonlinear Shell TSR6

Entity		Component													
Displacement	DX	DY	DZ	RSLT	THL1										
Stress	Nx	Ny	Nxy	Mx	My	Mxy	NMax	NMin	Ns	β	Nabs	Ne	Nx(T)/ Mx(T)	Ny(T)/ Ny(T)	Nx(B)/ Mx(B)
(continued)	Ny(B)/ My(B)	Util(T)	Util(B)	MUtil(T)	MUtil(B)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Fc(T)	Fc(B)
Eadp															
Strain	Ex	Ey	Exy	Bx	By	Bxy	EMax	EMin	EI	β	Eabs	EE			
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp								
Loading	FX	FY	FZ	RSLT	ML1										
Reaction	FX	FY	FZ	RSLT	ML1										
Residual Force	FX	FY	FZ	RSLT	ML1										
Reaction Stress	PX	PY	PZ												
Velocity	VX	VY	VZ	RSLT											
Acceleration	AX	AY	AZ	RSLT											
Plastic Strain															
Creep Strain															
Rubber Stretches															
TMB Stress	SX	SY	SZ	SXY	SYZ	SZX	S1	S2	S3	SI	Sabs	SE			
(continued)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	Eadp							
TMB Strain	EX	EY	EZ	EXY	EYZ	EZX	E1	E2	E3	EI	Eabs	EE			
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp								
TMB Plastic Strain	EPX	EPY	EPZ	EPXY	EPYZ	EPZX	EP1	EP2	EP3	EPI	EPabs	EPE	CWMax	EFSTMax	
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp								
TMB Creep Strain	ECX	ECY	ECZ	ECXY	ECYZ	ECZX	EC1	EC2	EC3	ECI	ECabs	ECE			
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp								

Notes:

TMB Plastic strain components CWMax and EFSTMax are only available when the Smoothed Multi-crack Concrete Model (Model 102) is used.

Semiloof Shells **TSL6**, **QSL8**

Entity	Component														
Displacement	DX	DY	DZ	RSLT	THL1	THL2									
Stress	Nx	Ny	Nxy	Mx	My	Mxy	NMax	NMin	Ns	β	Nabs	Ne	Nx(T)/ Mx(T)	Ny(T)/ My(T)	Nx(B)/ Mx(B)
(continued)	Ny(B)/ My(B)	Util(T)	Util(B)	MUtil(T)	MUtil(B)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Fc(T)	Fc(B)
(continued)	Eadp														
Strain	Ex	Ey	Exy	Bx	By	Bxy	EMax	EMin	EI	β	Eabs	EE			
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp								
Loading	FX	FY	FZ	RSLT	ML1	ML2									
Reaction	FX	FY	FZ	RSLT	ML1	ML2									
Residual Force	FX	FY	FZ	RSLT	ML1	ML2									
Reaction Stress	PX	PY	PZ												
Velocity	VX	VY	VZ	RSLT											
Acceleration	AX	AY	AZ	RSLT											
Plastic Strain															
Creep Strain															
Rubber Stretches															
TMB Stress	SX	SY	SZ	SXY	SYZ	SZX	S1	S2	S3	SI	Sabs	SE			
(continued)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp						
TMB Strain	EX	EY	EZ	EXY	EYZ	EXZ	E1	E2	E3	EI	Eabs	EE			
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp								
TMB Plastic Strain	EPX	EPY	EPZ	EPXY	EPYZ	EPZX	EP1	EP2	EP3	EPI	EPabs	EPE	CWMax	EFSMax	
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp								
TMB Creep Strain	ECX	ECY	ECZ	ECXY	ECYZ	ECZX	EC1	EC2	EC3	ECI	ECabs	ECE			
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp								

Notes

TMB Plastic strain components CWMax and EFSMax are only available when the Smoothed Multi-crack Concrete Model (Model 102) is used.

Thick Shells [TTS3](#), [TTS6](#), [QTS4](#), [QTS8](#)

Entity				Component												
Displacement	DX	DY	DZ	RSLT	THX	THY	THZ									
Stress	Nx	Ny	Nxy	Mx	My	Mxy	Sx	Sy	NMax	NMin	β	Nabs	NE	Nx(T)/ Mx(T)	Ny(T)/ My(T)	
(continued)	Nx(B)/Mx(B)	Ny(B)/ My(B)	Util(T)	Util(B)	MUtil(T)	MUtil(B)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Fc(T)	
(continued)	Fc(B)	Eadp														
Strain																
Loading	FX	FY	FZ	RSLT	MX	MY	MZ									
Reaction	FX	FY	FZ	RSLT	MX	MY	MZ									
Residual Force	FX	FY	FZ	RSLT	MX	MY	MZ									
Reaction Stress	PX	PY	PZ													
Velocity	VX	VY	VZ	RSLT												
Acceleration	AX	AY	AZ	RSLT												
Plastic Strain																
Creep Strain																
Rubber Stretches																
TMB Stress	SX	SY	SZ	SXY	SYZ	SZX	S1	S2	S3	SI	Nabs	SE				
(continued)	Damage	LogLife	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp							
TMB Strain	EX	EY	EZ	EXY	EYZ	EXZ	E1	E2	E3	EI	Eabs	EE				
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp									
TMB Plastic Strain	EPX	EPY	EPZ	EPXY	EPYZ	EPZX	EP1	EP2	EP3	EPI	EPabs	EPE	CWMax	EFSMax		
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp									
TMB Creep Strain	ECX	ECY	ECZ	ECXY	ECYZ	ECZX	EC1	EC2	EC3	ECI	ECabs	ECE				
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp									

Notes

Plastic strain components *CWMax* and *EFSMax* are only available when the Smoothed Multi-crack Concrete Model (Model 102) is used.

2D Joints (for Bars, Plane Stress and Plane Strain)

JNT3

Entity			Component					
Displacement	DX	DY	RSLT					
Stress	Fx	Fy	Damage	LogLife	SED	PWD	Eadp	
Strain	Ex	Ey	SED	PWD	Eadp			
Loading	FX	FY	RSLT					
Reaction	FX	FY	RSLT					
Residual Force	FX	FY	RSLT					
Reaction Stress								
Velocity	VX	VY	RSLT					
Acceleration	AX	AY	RSLT					
Plastic Strain	EPx	EPy	SED	PWD	Eadp			
Creep Strain								
Rubber Stretches								
TMB Stress								
TMB Strain								
TMB Plastic Strain								
TMB Creep Strain								

2D Joints (for Engineering and Kirchhoff Beams) **JPH3**

Entity	Component							
Displacement	DX	DY	RSLT	THZ				
Stress	Fx	Fy	Mz	Damage	LogLife	SED	PWD	Eadp
Strain	Ex	Ey	Bz	SED	PWD	Eadp		
Loading	FX	FY	RSLT	MZ				
Reaction	FX	FY	RSLT	MZ				
Residual Force	FX	FY	RSLT	MZ				
Reaction Stress								
Velocity	VX	VY	RSLT					
Acceleration	AX	AY	RSLT					
Plastic Strain	EPx	EPy	BPz	SED	PWD	Eadp		
Creep Strain								
Rubber Stretches								
TMB Stress								
TMB Strain								
TMB Plastic Strain								
TMB Creep Strain								

2D Joints (for Grillage Beams and Plates) **JF3**

Entity	Component							
Displacement	DZ	RSLT	THXZ	THY				
Stress	Fz	Mx	My	Damage	LogLife	SED	PWD	Eadp
Strain	Ez	Bx	By	SED	PWD	Eadp		
Loading	FZ	RSLT	MX	MY				
Reaction	FZ	RSLT	MX	MY				
Residual Force	FZ	RSLT	MX	MY				
Reaction Stress								
Velocity	VZ	RSLT						
Acceleration	AZ	RSLT						
Plastic Strain	EPx	EPy	BPz	SED	PWD	Eadp		
Creep Strain								
Rubber Stretches								
TMB Stress								
TMB Strain								
TMB Plastic Strain								
TMB Creep Strain								

2D Joints (for Axisymmetric Solids) [**JAX3**](#)

Entity	Component						
Displacement	DX	DY	RSLT				
Stress	Fx	Fy	Damage	LogLife	SED	PWD	Eadp
Strain	Ex	Ey	SED	PWD	Eadp		
Loading	FX	FY	RSLT	MZ			
Reaction	FX	FY	RSLT	MZ			
Residual Force	FX	FY	RSLT	MZ			
Reaction Stress							
Velocity	VX	VY	RSLT				
Acceleration	AX	AY	RSLT				
Plastic Strain	EPx	EPy	SED	PWD	Eadp		
Creep Strain							
Rubber Stretches							
TMB Stress							
TMB Strain							
TMB Plastic Strain							
TMB Creep Strain							

2D Joints (for Axisymmetric Shells) **JXS3**

Entity	Component							
Displacement	DX	DY	RSLT	THZ				
Stress	Fx	Fy	Mz	Damage	LogLife	SED	PWD	Eadp
Strain	Ex	Ey	Bz	SED	PWD	Eadp		
Loading	FX	FY	RSLT	MZ				
Reaction	FX	FY	RSLT	MZ				
Residual Force	FX	FY	RSLT	MZ				
Reaction Stress								
Velocity	VX	VY	RSLT					
Acceleration	AX	AY	RSLT					
Plastic Strain	EPx	EPy	BPz	SED	PWD	Eadp		
Creep Strain								
Rubber Stretches								
TMB Stress								
TMB Strain								
TMB Plastic Strain								
TMB Creep Strain								

3D Joints (for general 3 dof connection) **JNT4**, **JL43**

(for Bars, Solids, Space Membranes and Semiloof Shell Corners)

Entity				Component				
Displacement	DX	DY	DZ	RSLT				
Stress	Fx	Fy	Fz	Damage	LogLife	SED	PWD	Eadp
Strain	Ex	Ey	Ez	SED	PWD	Eadp		
Loading	FX	FY	FZ	RSLT				
Reaction	FX	FY	FZ	RSLT				
Residual Force	FX	FY	FZ	RSLT				
Reaction Stress								
Velocity	VX	VY	VZ	RSLT				
Acceleration	AX	AY	AZ	RSLT				
Plastic Strain	EPx	EPy	EPz	SED	PWD	Eadp		
Creep Strain								
Rubber Stretches								
TMB Stress								
TMB Strain								
TMB Plastic Strain								
TMB Creep Strain								

3D Joints (for general 6 dof connection) [JSH4](#), [JL46](#)

(for Engineering, Kirchhoff and Semiloof Beam End Nodes)

Entity		Component										
Displacement		DX	DY	DZ	RSLT	THX	THY	THZ				
Stress		Fx	Fy	Fz	Mx	My	Mz	Damage	LogLife	SED	PWD	Eadp
Strain		Ex	Ey	Ez	Bx	By	Bz	SED	PWD	Eadp		
Loading		FX	FY	FZ	RSLT	MX	MY	MZ				
Reaction		FX	FY	FZ	RSLT	MX	MY	MZ				
Residual Force		FX	FY	FZ	RSLT	MX	MY	MZ				
Reaction Stress												
Velocity		VX	VY	VZ	RSLT							
Acceleration		AX	AY	AZ	RSLT							
Plastic Strain		EPx	EPy	EPz	BPx	BPy	BPz	SED	PWD	Eadp		
Creep Strain												
Rubber Stretches												
TMB Stress												
TMB Strain												
TMB Plastic Strain												
TMB Creep Strain												

3D Joints (for Semiloof Element Mid-side Nodes) [**JSL4**](#)

Entity				Component						
Displacement	DX	DY	DZ	RSLT	THL1	THL2				
Stress	Fx	Fy	Fz	M1	M2	Damage	LogLife	SED	PWD	Eadp
Strain	Ex	Ey	Ez	B1	B2	SED	PWD	Eadp		
Loading	FX	FY	FZ	RSLT	ML1	ML2				
Reaction	FX	FY	FZ	RSLT	ML1	ML2				
Residual Force	FX	FY	FZ	RSLT	ML1	ML2				
Reaction Stress										
Velocity	VX	VY	VZ	RSLT						
Acceleration	AX	AY	AZ	RSLT						
Plastic Strain	EPx	EPy	EPz	BP1	BP2	SED	PWD	Eadp		
Creep Strain										
Rubber Stretches										
TMB Stress										
TMB Strain										
TMB Plastic Strain										
TMB Creep Strain										

Thermal Bars [BFD2/3](#), [BFS2/3](#), [BFX2/3](#)

Entity	Component
Potential	PHI
Gradient	Gx Eadp
Flux	qx Eadp
Reaction	Q

Thermal Links [LFD2](#), [LFS2](#), [LFX2](#)

Entity	Component
Potential	PHI
Gradient	Gx Eadp
Flux	qx Eadp
Reaction	Q

Plane and Axisymmetric Field [TFD3/6](#), [QFD4/8](#), [TXF3/6](#), [QXF4/8](#)

Entity	Component
Potential	PHI
Gradient	Gx Gy Eadp
Flux	qx qy Eadp
Reaction	Q

Solid Field [**TF4/10**](#), [**PF6/12/15**](#), [**HF8/16/20**](#), [**TF10S**](#),
[**PF6C/12C**](#), [**HF8C/16C**](#)

Entity	Component			
Potential	PHI			
Gradient	Gx	Gy	Gz	Eadp
Flux	qx	qy	qz	Eadp
Reaction	Q			

2D Interface Element [IPN4](#), [IPN6](#), [IAX4](#), [IAX6](#)

Entity	Component					
Displacement	Dx	Dy	RSLT			
Stress	Sx	Sy	Damage	LogLife	Eadp	
Strain	Ex	Ey	Eadp			
Loading	Fx	Fy	RSLT	MZ		
Reaction	Fx	Fy	RSLT	MZ		
Residual Force	Fx	Fy	RSLT			
Reaction Stress						
Velocity	Vx	Vy	RSLT			
Acceleration	Ax	Ay	RSLT			
Plastic Strain						
Creep Strain						
Rubber Stretches						
TMB Stress						
TMB Strain						
TMB Plastic Strain						
TMB Creep Strain						

3D Interface Element [IS6](#), [IS8](#), [IS12](#), [IS16](#)

Entity	Component					
Displacement	Dx	Dy	RSLT			
Stress	Sx	Sy	Sz	Damage	LogLife	Eadp
Strain	Ex	Ey	Eadp			
Loading	Fx	Fy	Fz	RSLT		
Reaction	Fx	Fy	Fz	RSLT		
Residual Force	Fx	Fy	Fz	RSLT		
Reaction Stress						
Velocity	Vx	Vy	Vz	RSLT		
Acceleration	Ax	Ay	Az	RSLT		
Plastic Strain						
Creep Strain						
Rubber Stretches						
TMB Stress						
TMB Strain						
TMB Plastic Strain						
TMB Creep Strain						

Appendix L : Joint Element Compatibility.

Joint Element Compatibility

Joint elements are compatible with the following elements:

Joint Element		Compatible Finite Elements
JNT3	Bars	BAR2, BAR3
	2D Plane Stress	QPM4, QPM8, TPM3, TPM6, QPK8, TPK6, QPM4M
	2D Plane Strain	QPN4, QPN8, QPN8P, TPN3, TPN6, TPN6P, QNK8, TNK6, QPN4M, QPN4L
JPH3	2D Beams	BEAM, BM3, BMX3
JF3	2D Grillage	GRIL
	2D Plates	TF3, QF4, TF6, QSC4, TTF6, QTF8
JNT4	3D Bars	BRS2, BRS3
	3D Solids	HX8, HX16, HX16P, HX20, HX20P, PN6, PN12, PN12P, PN15, PN15P, TH4, TH10, HX8M, HX8L, HX16L, PN6L, PN12L, TH10S
	Space Membranes	TSM3, SMI4
	3D Shell	TSR6 (corner nodes)

Joint Element	Compatible Finite Elements	
JL43	Semiloof Shells	TSL6, QSL8 (corner nodes)
JSH4	3D Beams	BMS3, BTS3, BS3, BS4, BSX4, BMI21, BMI22, BMI31, BMI33, BMX21, BMX31, BMX22, BMX33
	3D Shells	TS3, QSI4, TTS3, TTS6, QTS4, QTS8
JL46	Semiloof Beams	BSL3, BSL4, BXL4 (corner nodes)
JSL4	Semiloof Beams	BSL3, BSL4, BXL4 (mid-side nodes)
	Semiloof Shells	QSL8, TSL6 (mid-side nodes)
JAX3	Axisymmetric Solids	QAX4, QAX8, QZX8P, TAX3, TAX6, TAX6P, TXK6, QXK8, QAX4M, QAX4L
JXS3	Axisymmetric Shells	BXS3

Notes on the use of Joints

1. The nodes of a joint element need not be coincident, but for correct response the distance between them should be as small as possible. This is particularly important with joint elements which contain rotational degrees of freedom, since the stiffness matrix is not formulated using engineering beam theory. This means that a joint moment is independent of both shear force and its length. For instance, the moment calculated with a joint length of zero will remain the same magnitude at any other joint length. These effects can be exacerbated significantly in dynamic analyses (e.g. eigenvalue extraction or Hilber dynamics). Non-coincident nodes will lead to additional forces in the solution which are not in equilibrium (usually small and swamped, but could be significant sometimes). It is not recommend to have joints “hanging off” the side of a modelled structure, having a large stiffness associated.
2. Joints do not support any geometric nonlinearity. They may be used, however, in geometrically nonlinear analyses but will themselves remain geometrically linear (that is, infinitesimal strain is assumed and large deformation effects are ignored).
3. The strain for a joint element is measured as follows:
 - Strain measure = (displacement for 2nd node) - (displacement for 1st node)
 - This strain being measured in the local axis system. Therefore, if node 1 is restrained, node 2 would need to be displaced in the negative local (x/y/z) direction to generate compressive contact forces.
4. The rotation output for a joint element is measured in radians.

Index

2

2D Continuum, 149, 155, 161, 166,
171, 177, 183, 187, 193, 198, 203, 209,
215, 220, 226, 231, 237
2D Interface, 443
2D Joints, 359, 364, 368, 372
2D Line, 462
2D Point, 453
2D Rigid Surface, 471

3

3D Continuum, 245, 252, 258, 264,
270, 276
3D Interface, 447
3D Joints, 384, 388
3D Line, 459
3D Point, 456
3D Rigid Surface, 474
3D Surface, 465

A

arch, 113, 120, 126
axial force, 62
Axisymmetric Shells, 305

Axisymmetric Solid, 203, 209, 215,
220
Axisymmetric Solid Continuum, 226
Axisymmetric Solid Two-Phase, 231

B

BAR2, 59, 527
BAR3, 59, 527
Bars, 59, 64
BEAM, 71, 529
Beam Elements, 71, 76, 82, 91, 99,
106, 108, 114, 121, 127, 134, 140
Beams, 108, 114
BFD2, 395, 569
BFD3, 395, 569
BFS2, 403, 569
BFS3, 403, 569
BFX2, 399, 569
BFX3, 399, 569
BM3, 108, 534
BMI21, 93
BMI22, 93
BMI31, 93
BMI32, 93
BMS3, 76, 530
BMX21, 101
BMX22, 101
BMX3, 114, 534
BMX31, 101

BMX33, 101
BRS2, 64, 528
BRS3, 64, 528
BS3, 121, 535
BS4, 121, 535
BSL3, 134, 533, 536
BSL4, 134, 533, 536
BSX4, 127, 535
BTS3, 86, 532
BXL4, 140, 533, 536
BXM2, 343, 554
BXM3, 343, 554
BXS3, 305, 556

C

cable structures, 64
cables, 62, 67, 80, 91
circular plates, 347
composite, 425, 430

D

delamination, 443

E

Element Loads, 477
Engineering Beam Elements, 493, 494
Engineering Beams, 71, 76, 82
Environmental Temperature Loading, 480
Excessive Aspect Ratios, 487
Excessive Element Curvature, 487
Excessive Warping, 488

Explicit Dynamics, 538, 542, 543

F

Face loading, 481
 Face Loads On 2D Continuum Elements, 482
 Face Loads On 3D Continuum Elements, 483
 For Thermal Bars, 481
Field, 395, 399, 403, 407, 410, 413, 416, 420, 425, 430, 435
Flat Thin Shells, 311, 317
Fourier Ring, 237
fracture mechanics, 192
frame, 80, 111, 118, 125, 131, 139, 145

G

GRIL, 82, 531
grillage, 82
Grillage Elements, 492
groundwater, 439

H

heat conduction, 398, 409, 415
HF16, 420, 570
HF16C, 430, 570
HF20, 420, 570
HF8, 420, 570
HF8C, 430, 570
HX16, 245, 548
HX16L, 270, 548
HX16P, 281, 549

HX20, 245, 548
HX20K, 258
HX20P, 281, 549
HX8, 245, 548
HX8E, 550
HX8L, 270, 548
HX8M, 252, 548

I

IAx4, 443, 571
IAx6, 443
Interface, 443, 447
IPN4, 443, 571
IPN6, 443, 571
IS12, 447, 572
IS16, 447, 572
IS6, 447, 572
IS8, 447, 572
Isoflex Plates, 289, 293

J

JAX3, 368, 564
JF3, 364, 563
JL43, 380, 566
JL46, 384, 567
JNT3, 355, 561
JNT4, 376, 566
Joint Element Compatibility, 573
Joints, 359, 364, 368, 372, 384, 388, 574
JPH3, 359, 562
JSH4, 384, 567
JSL4, 388, 568

JXS3, 372, 565

K

Kirchhoff Beams, 108, 114, 121, 127

L

LFD2, 407, 569
LFS2, 410, 569
LFX2, 413, 569
LM2, 462
LM3, 462
LMS3, 459
LMS4, 459
Load types, 480
LUSAS Element Types, 12

M

Mass Lumping in LUSAS, 513
Membranes, 348
Mid-side Node Centrality, 487
Mindlin Plates, 297
modelling reinforcement, 67
Moments of Inertia Definitions, 515

N

Newton-Cotes Integration Points, 505
Non-Structural Mass, 453, 456, 459, 462, 465
Numerically Integrated Beam Elements, 491

O

Output Notation for Principal Stresses, 511
Overview, 7

P

perforated thick plates, 296
PF12, 420, 570
PF12C, 430, 570
PF15, 420, 570
PF6, 420, 570
PF6C, 430, 570
pipes, 310, 347
Plane Field, 416, 435
plane frames, 111, 118
Plane Strain Continuum, 171, 177, 183, 187, 193, 198
Plane Stress Continuum, 149, 155, 161, 166
Plates, 289, 293, 297
PM2, 453
PM3, 456
PN12, 245, 548
PN12L, 270, 548
PN12P, 281, 549
PN15, 245, 548
PN15K, 258
PN15P, 281, 549
PN6, 245, 548
PN6E, 276, 550
PN6L, 270, 548
pressure vessels, 310

Q

QAX4, 203, 546
QAX4E, 226, 543
QAX4F, 237, 545
QAX4L, 215, 547
QAX4M, 209, 546
QAX8, 203, 231, 546
QAX8F, 237, 545
QF4, 289, 551
QFD4, 416, 569
QFD8, 416, 569
QM4, 465
QM8, 465
QNK8, 187, 539
QPK8, 161, 537
QPM4, 149, 537
QPM4E, 166, 538
QPM4M, 155, 537
QPM8, 149, 537
QPN4, 171, 539
QPN4E, 193, 542
QPN4L, 183, 540
QPN4M, 177, 539
QPN8, 171, 539
QPN8P, 198, 541, 544
QSC4, 293, 552
QSI4, 311, 557
QSL8, 323, 559
QTF8, 297, 553
QTS4, 331, 560
QTS8, 331, 560
QXF4, 435, 569
QXF8, 435, 569
QXK8, 220, 546

R

R2D2, 471
R3D3, 474
R3D4, 474
reinforced concrete, 67
Results Notation
 Key to Results Tables, 517
Rigid, 471, 474
Rigid Surface 3D Elements, 474

S

SED, 524
Semiloof Beams, 134, 140
Semiloof Shells, 323
Shear Areas, 507
shell structures, 330
Shells, 305, 311, 317, 323, 331
SMI4, 348, 555
Solid Continuum, 245, 252, 258, 264, 270, 276
Solid Continuum Crack Tip, 258
Solid Field, 420, 425, 430
space frames, 126, 133
space frames., 145
Space Membranes, 348
Standard 2D Continuum Element, 496
Standard 3D Continuum Element, 496
Standard Bar Element, 491
Standard Beam Eccentricity, 496
Standard Field Element, 500
Standard Joint Element, 489, 500
Standard Line Element, 489
Standard Membrane Element, 500
Standard Plate Element, 497

Standard Surface Element, 489
Strain energy density, 524
Structural Bars, 59, 64
Surface Mass Elements, 465

T

TAX3, 203, 546
TAX3E, 226, 543
TAX3F, 237, 545
TAX6, 203, 546
TAX6F, 237, 545
TAX6P, 231
temperature, 399
temperature distribution, 439
Tetrahedral, 264
TF10, 420, 570
TF10S, 425, 570
TF3, 289, 551
TF4, 420, 570
TFD3, 416, 569
TFD6, 416, 569
TH10, 245, 548
TH10K, 258
TH10P, 281
TH10S, 264, 548
TH4, 245, 548
TH4E, 276, 550
thermal analysis, 419
Thermal Bars, 395, 399, 403
Thermal Links, 407, 410, 413
Thick Shell Eccentricity, 499
Thick Shell Element, 499
Thick Shell Nodal Rotation, 501
Thick Shells, 331, 560
Thin Shell Eccentricity, 498

Thin Shell Element, 497	TS3, 311, 557
TM3, 465	TSL6, 323, 559
TM6, 465	TSM3, 348, 555
TNH10P, 549	TSR6, 317, 558
TNK6, 187, 539	TTF6, 297, 553
Torsional Constant, 508	TTS3, 331, 560
TPK6, 161, 537	TTS6, 331, 560
TPM3, 149, 537	two phase, 281
TPM3E, 166, 538	TXF3, 435, 569
TPM6, 149, 537	TXF6, 435, 569
TPN3, 171, 539	TXK6, 220, 546
TPN3E, 193, 542	
TPN6, 171, 539	
TPN6P, 198, 541, 544	
Transforming Results Directions, 525	
trusses, 67	

U

UDL Loads on Shells, 485