Element Reference Manual

Version 15.1 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

Table of Contents

Notation.

A	Cross sectional area
Ap	Plastic area
As, Asy, Asz	Effective shear area
$\mathbf{A}_1$ **...** $\mathbf{A}_n$	Nodal cross sectional areas
ar	Mass Rayleigh damping constant
α	Coefficient of thermal expansion
α**s**	Softening parameter
α**x**, α**y**, α**z**, α**xy**, α**xz**, α**yz**	Orthotropic thermal expansion coefficients
α**x**, α**y**, α**z**	Angular accelerations
br	Stiffness Rayleigh damping parameter
β	Shear retention factor/parameter
β	Principal stresses direction
C	Specific heat capacity
Ci	(i)th hardening stiffness
$\mathbf{C}_0$	Neo-Hookean rubber model constant
$\mathbf{C}_1$, $\mathbf{C}_2$	Mooney-Rivlin rubber model constants
c	Cohesion
co	Initial cohesion
Dij	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

ε**x**, ε**y**, ε**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

ψ**xy**, ψ**xz**, ψ**yz** Torsional strain resultants

G Shear modulus

Gf Fracture energy

Gxy, Gxz, Gyz Orthotropic shear moduli

γ**x**, γ**y**, γ**z** Membrane strain resultants

γ**x**, γ**y**, γ**z** Field gradients (local or global)

H Enthalpy

Hi1 Isotropic hardening parameter

Hk1 Kinematic hardening parameter

hc Convective heat transfer coefficient

hf Heat fraction

hr Radiative heat transfer coefficient

θ**x**, θ**y**, θ**z**	Rotations (local or global)
θ_1, θ_2	Loof node rotations (local)
$\theta\alpha$, $\theta\beta$	Nodal rotations for thick shells
$\theta\lambda$	Angle defining principal directions of λ_1, λ_2
Iy, Iz	1st moments of inertia
Iyy, Izz	2nd moments of inertia
Iyz	Product moment of inertia
J	Volume ratio (determinant of F)
K	Spring stiffness
Kc	Contact stiffness
Kl	Lift-off stiffness
Ko	Original gap conductance
Kt	Torsional constant
k	Thermal conductivity
kx, ky, kz	Orthotropic thermal conductivities
kr	Bulk modulus
κ	Hardening stiffness
Li	Limit of (i)th hardening stiffness
λ_1, λ_2, λ_3	Principal stretches
M	Mass
Mx, My, Mz	Concentrated moments (local or global)
Mx, My, Mz, $M\theta$	Flexural moments (local or global)
Mxy, Mxz, Myz	Torsional moments (local or global)
M_1, M_2	Concentrated loof moments (local or global)
m_x, m_y, m_z	Mass in element local directions
μ	Coulomb friction coefficient
μ**ri**, α**ri**	Ogden rubber model constants

Nx, Ny, Nz, Nθ	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
υ	Poisson's ratio
υ**xy**, υ**xz**, υ**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**yo**	Initial uniaxial yield stress
σ**x**, σ**y**, σ**z**	Direct stresses (local or global)
σ**max**, σ**min**	Principal stresses
σ**xy**, σ**xz**, σ**yz**	Shear stresses (local or global)
σ**s**	Maximum shear stress
σ**e**	Von Mises equivalent stress
T	Temperature
T, To	Final, initial temperatures
$t_1 \ldots t_n$	Nodal thicknesses
U, V, W	Displacements (global)
Φ	Field variable

Φe	External environmental temperature
ϕ	Frictional angle
ϕo	Initial frictional angle
ϕ	Body force potential
Vx, Vy, Vz	Nodal velocities (global)
V11, V12 ... V33	Left stretch tensor components
Wx, Wy, Wz	Uniformly distributed intensities
X, Y, Z	Nodal coordinates (global)
Xcbf, Ycbf, Zcbf	Constant body forces (global)
Xo, Yo, Zo	Offsets of finite element model coordinate system from point about which global angular acceleration and velocities are applied
$y_1, z_1 \ldots y_4, z_4$	Cross sectional coordinates (local)
Zy_p, Zz_p	Torsional plastic moduli
Zyy_p, Zzz_p	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

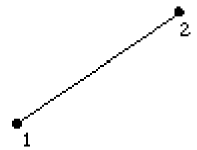
Of these three methods, the element summary tables provide the most detail to enable correct element selection for a particular modelling task.

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**

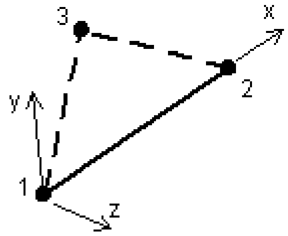
Bar Elements



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.

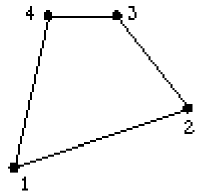
Beam Elements



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***.

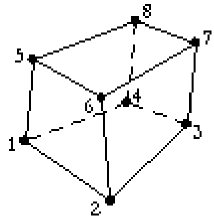
2D Continuum Elements



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

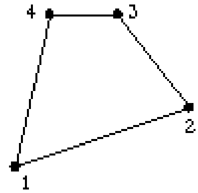
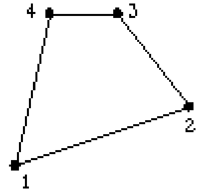


Plate elements are used to model flat structures whose deformation can be assumed to be predominantly flexural

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

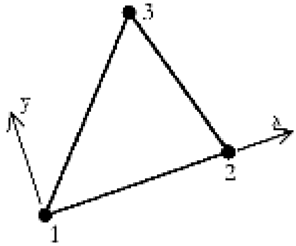
Shell Elements



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.

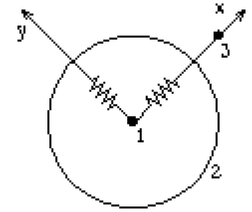
Membrane Elements



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).

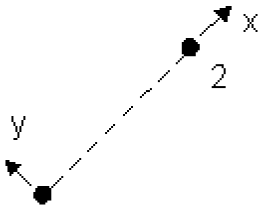
Joint Elements



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.

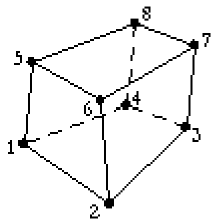
Non-Structural Mass Elements



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.

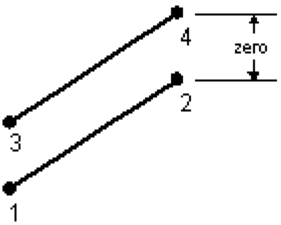
Field Elements



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.

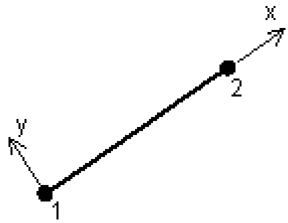
Interface Elements



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, QPM4, QPM8, QPM4M, TPK6, QPK8	TPM3E, QPM4E
	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	
	Flat thin shells		TS3, QSI4	TSR6,

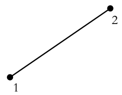
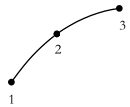
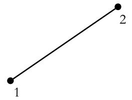
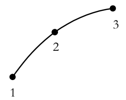
Element Group	Element Subgroup	Element Name and Software Product Version Availability		
		LT	Standard	Plus
	Semiloof shells			TSL6, QSL8
	Thick shells		TTS3, QTS4	TTS6, QTS8
Membranes	Axisymmetric membranes		BXM2, BXM3	
	Space membranes		TSM3, SMI4	
Joints	2D joints		JNT3, JPH3, JF3, JAX3, JXS3	
	3D 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

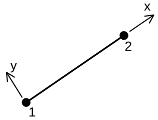
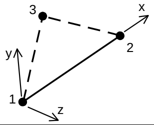
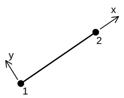
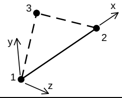
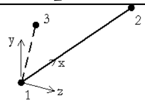
The following element index tables provide a diagrammatic index for each element with a description of the element, the nodal freedoms, and the software product version in which it is available.

The tables are listed in the following order:

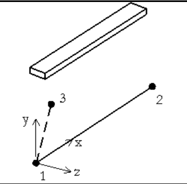
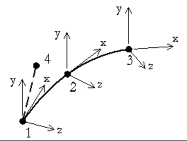
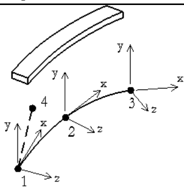
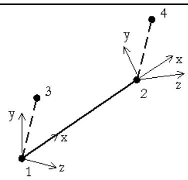
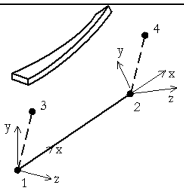
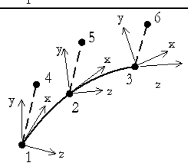
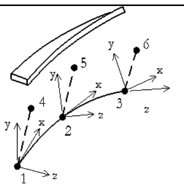
Bar Elements

Name	Geometry	Title	Freedoms	Product Version
BAR2		BAR element in 2D	U, V	LT
BAR3		BAR element in 2D	U, V	Standard
BRS2		BAR element in 3D	U, V, W	LT
BRS3		BAR element in 3D	U, V, W	Standard

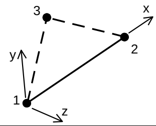
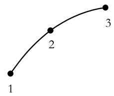
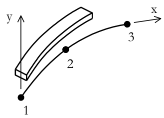
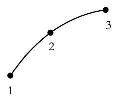
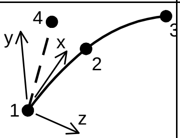
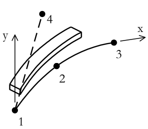
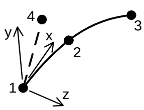
Beam Elements

Name	Geometry	Title	Freedoms	Product Version
BEAM		ENGINEERING thick beam element in 2D	U, V, qz	LT
BMS3		ENGINEERING thick beam element in 3D	U, V, W, qx, qy, qz	LT
GRIL		ENGINEERING grillage thick beam element in 2D	W, qx, qy	LT
BTS3		THICK beam element in 3D (co-rotational)	U, V, W, qx, qy, qz	Standard
BMI21		THICK linear thick beam element in 3D	U, V, W, qx, qy, qz	LT

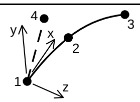
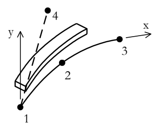
Beam Elements

Name	Geometry	Title	Freedoms	Product Version
BMX21		THICK linear thick beam element in 3D with quadrilateral cross-section	U, V, W, qx, qy, qz	Standard
BMI31		THICK quadratic thick beam element in 3D	U, V, W, qx, qy, qz	Plus
BMX31		THICK quadratic thick beam element in 3D with quadrilateral cross-section	U, V, W, qx, qy, qz	Plus
BMI22		THICK twisted linear thick beam element in 3D	U, V, W, qx, qy, qz	Plus
BMX32		THICK twisted linear thick beam element in 3D with quadrilateral cross-section	U, V, W, qx, qy, qz	Plus
BMI33		THICK twisted quadratic thick beam element in 3D with quadrilateral cross-section	U, V, W, qx, qy, qz	Plus
BMX33		THICK beam element in 3D	U, V, W, qx, qy, qz	Plus

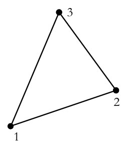
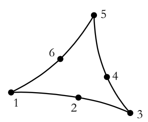
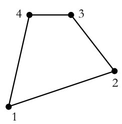
Beam Elements

Name	Geometry	Title	Freedoms	Product Version
BTS3		THICK beam element in 3D (co-rotational)	U, V, W, qx, qy, qz	Standard
BM3		KIRCHHOFF thin beam element in 2D	end nodes: U, V, qz mid-node: dU	Standard
BMX3		KIRCHHOFF thin beam element in 2D with quadrilateral cross-section	end nodes: U, V, qz mid-node: dU	Standard
BS3		KIRCHHOFF thin beam element in 3D	end nodes: U, V, W, qx, qy, qz mid-node: dU, dqx	Plus
BS4		KIRCHHOFF thin beam element in 3D	end nodes: U, V, W, qx, qy, qz mid-node: dU, dqx	Plus
BSX4		KIRCHHOFF thin beam element in 3D with quadrilateral cross-section	end nodes: U, V, W, qx, qy, qz mid-node: dU, dqx	Plus
BSL3		SEMILOOF thin beam element in 3D for use with TSL6	end nodes: U, V, W,	Plus

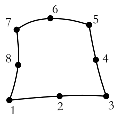
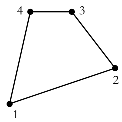
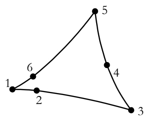
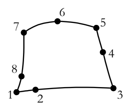
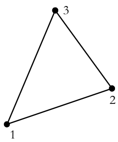
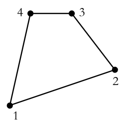
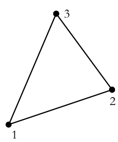
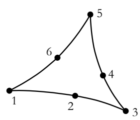
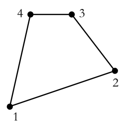
Beam Elements

Name	Geometry	Title	Freedoms	Product Version
			qx, qy, qz mid- node: U, V, W, q1, q2	
BSL4		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
BXL4		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

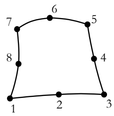
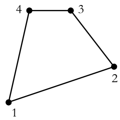
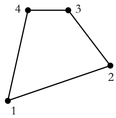
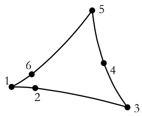
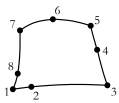
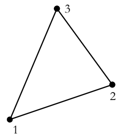
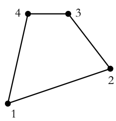
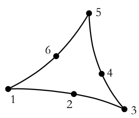
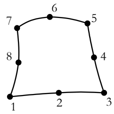
2D Continuum Elements

Name	Geometry	Title	Freedoms	Product Version
TPM3		PLANE STRESS continuum element in 2D	U, V	Standard
TPM6		PLANE STRESS continuum element in 2D	U, V	Standard
QPM4		PLANE STRESS continuum element in 2D	U, V	Standard

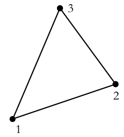
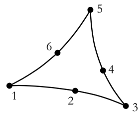
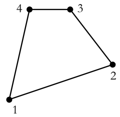
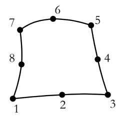
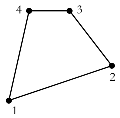
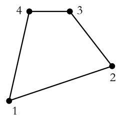
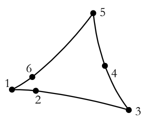
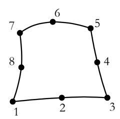
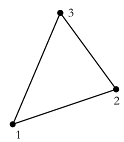
2D Continuum Elements

Name	Geometry	Title	Freedoms	Product Version
QPM8		PLANE STRESS continuum element in 2D	U, V	Standard
QPM4M		PLANE STRESS continuum element in 2D with enhanced strains	U, V	Standard
TPK6		PLANE STRESS continuum crack tip element in 2D	U, V	Standard
QPK8		PLANE STRESS continuum crack tip element in 2D	U, V	Standard
TPM3E		PLANE STRESS explicit dynamics element in 2D	U, V	Plus
QPM4E		PLANE STRESS explicit dynamics element in 2D	U, V	Plus
TPN3		PLANE STRAIN continuum element in 2D	U, V	Standard
TPN6		PLANE STRAIN continuum element in 2D	U, V	Standard
QPN4		PLANE STRAIN continuum element in 2D	U, V	Standard

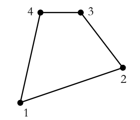
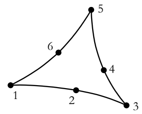
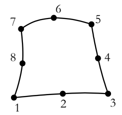
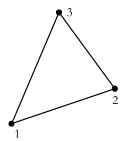
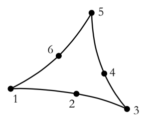
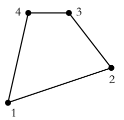
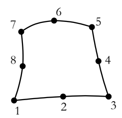
2D Continuum Elements

Name	Geometry	Title	Freedoms	Product Version
QPN8		PLANE STRAIN continuum element in 2D	U, V	Standard
QPN4M		PLANE STRAIN continuum element in 2D with enhanced strains	U, V	Standard
QPN4L		PLANE STRAIN continuum element in 2D for large strains	U, V	Standard
TNK6		PLANE STRAIN continuum crack tip element in 2D	U, V	Standard
QNK8		PLANE STRAIN continuum crack tip element in 2D	U, V	Standard
TPN3E		PLANE STRAIN explicit dynamics element in 2D	U, V	Plus
QPN4E		PLANE STRAIN explicit dynamics element in 2D	U, V	Plus
TPN6P		PLANE STRAIN continuum two phase element in 2D	U, V P: corner nodes U, V: Midside nodes	Standard
QPN8P		PLANE STRAIN continuum two phase element in 2D	U, V P: corner nodes U,	Standard

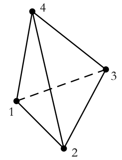
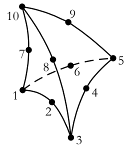
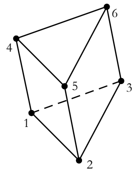
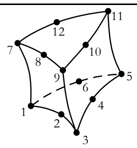
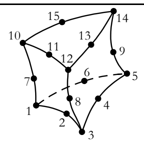
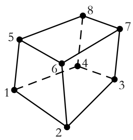
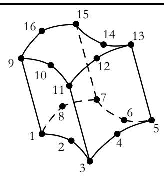
2D Continuum Elements

Name	Geometry	Title	Freedoms	Product Version
			V: Midsi de nodes	
TAX3		AXISYMMETRIC solid continuum element in 2D	U, V	Standard
TAX6		AXISYMMETRIC solid continuum element in 2D	U, V	Standard
QAX4		AXISYMMETRIC solid continuum element in 2D	U, V	Standard
QAX8		AXISYMMETRIC solid continuum element in 2D	U, V	Standard
QAX4M		AXISYMMETRIC solid continuum element in 2D with enhanced strains	U, V	Standard
QAX4L		AXISYMMETRIC solid continuum element in 2D for large strains	U, V	Standard
TXK6		AXISYMMETRIC solid continuum crack tip element in 2D	U, V	Standard
QXK8		AXISYMMETRIC solid continuum crack tip element in 2D	U, V	Standard
TAX3E		AXISYMMETRIC solid explicit dynamics element in 2D	U, V	Plus

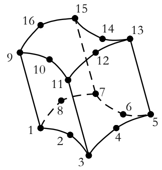
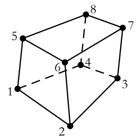
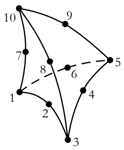
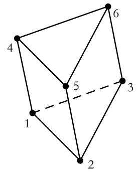
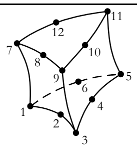
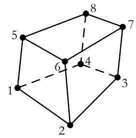
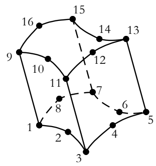
2D Continuum Elements

Name	Geometry	Title	Freedoms	Product Version
QAX4E		AXISYMMETRIC solid explicit dynamics element in 2D	U, V	Plus
TAX6P		AXISYMMETRIC solid two phase continuum element in 2D	U, V P: corner nodes U, V: Midside nodes	Plus
QAX8P		AXISYMMETRIC solid two phase continuum element in 2D	U, V P: corner nodes U, V: Midside nodes	Plus
TAX3F		AXISYMMETRIC Fourier ring element in 2D	U, V, W	Plus
TAX6F		AXISYMMETRIC Fourier ring element in 2D	U, V, W	Plus
QAX4F		AXISYMMETRIC Fourier ring element in 2D	U, V, W	Plus
QAX8F		AXISYMMETRIC Fourier ring element in 2D	U, V, W	Plus

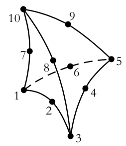
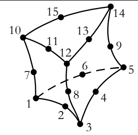
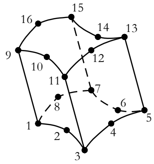
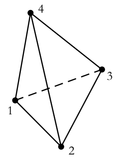
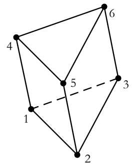
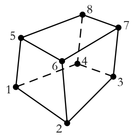
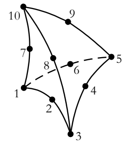
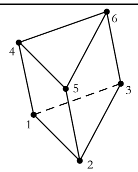
3D Continuum Elements

Name	Geometry	Title	Freedoms	Product Version
TH4		SOLID CONTINUUM element in 3D	U, V, W	Standard
TH10		SOLID CONTINUUM element in 3D	U, V, W	Plus
PN6		SOLID CONTINUUM element in 3D	U, V, W	Standard
PN12		SOLID CONTINUUM element in 3D	U, V, W	Plus
PN15		SOLID CONTINUUM element in 3D	U, V, W	Plus
HX8		SOLID CONTINUUM element in 3D	U, V, W	Standard
HX16		SOLID CONTINUUM element in 3D	U, V, W	Plus

3D Continuum Elements

Name	Geometry	Title	Freedoms	Product Version
HX20		SOLID CONTINUUM element in 3D	U, V, W	Plus
HX8M		SOLID CONTINUUM element in 3D with enhanced strains	U, V, W	Standard
TH10S		SOLID CONTINUUM composite element in 3D	U, V, W	Plus
PN6L		SOLID CONTINUUM composite element in 3D	U, V, W	Plus
PN12L		SOLID CONTINUUM composite element in 3D	U, V, W	Plus
HX8L		SOLID CONTINUUM composite element in 3D	U, V, W	Plus
HX16L		SOLID CONTINUUM composite element in 3D	U, V, W	Plus

3D Continuum Elements

Name	Geometry	Title	Freedoms	Product Version
TH10K		SOLID CONTINUUM crack tip element in 3D	U, V, W	Plus
PN15K		SOLID CONTINUUM crack tip element in 3D	U, V, W	Plus
HX20K		SOLID CONTINUUM crack tip element in 3D	U, V, W	Plus
TH4E		SOLID CONTINUUM explicit dynamics element in 3D	U, V, W	Plus
PN6E		SOLID CONTINUUM explicit dynamics element in 3D	U, V, W	Plus
HX8E		SOLID CONTINUUM explicit dynamics element in 3D	U, V, W	Plus
TH10P		SOLID CONTINUUM two phase element in 3D	U, V, W	Plus
PN12P		SOLID CONTINUUM two phase element in 3D	U, V, W	Plus

3D Continuum Elements

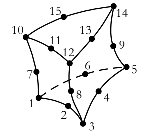
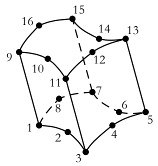
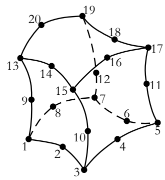
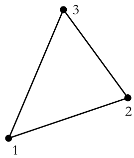
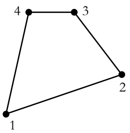
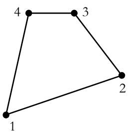
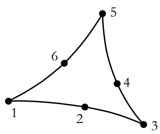
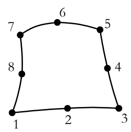
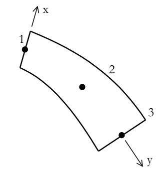
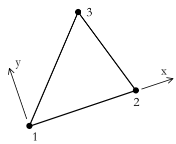
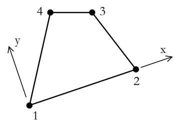
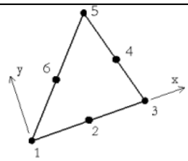
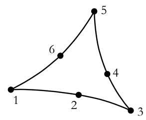
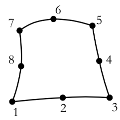
Name	Geometry	Title	Freedoms	Product Version
PN15P		SOLID CONTINUUM two phase element in 3D	U, V, W	Plus
HX16P		SOLID CONTINUUM two phase element in 3D	U, V, W	Plus
HX20P		SOLID CONTINUUM two phase element in 3D	U, V, W	Plus

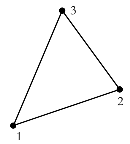
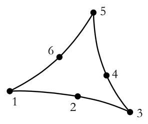
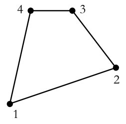
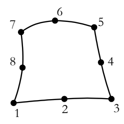
Plate Elements

Name	Geometry	Title	Freedoms	Product Version
TF3		ISOFLEX thin plate flexure element in 2D	W, qx, qy	Standard
QF4		ISOFLEX thin plate flexure element in 2D	W, qx, qy	Standard
QSC4		ISOFLEX thick plate flexure element in 2D	W, qx, qy	Standard
TTF6		MINDLIN thick plate flexure element in 2D	W, qx, qy	Standard
QTF8		MINDLIN thick plate flexure element in 2D	W, qx, qy	Standard

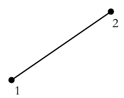
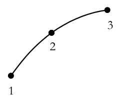
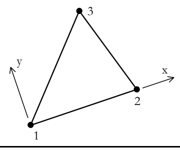
Shell Elements

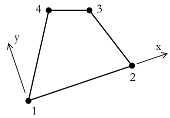
Name	Geometry	Title	Freedoms	Product Version
BXS3		AXISYMMETRIC thin shell element in 2D	end nodes: U, V, qz mid-node: dU	Standard
TS3		FLAT thin shell element in 3D	U, V, W, qx, qy, qz	Standard
QSI4		FLAT thin shell element in 3D	U, V, W, qx, qy, qz	Standard
TSR6		FLAT thin nonlinear shell element in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

Shell Elements

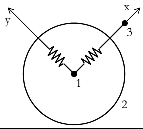
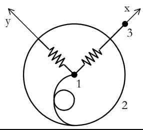
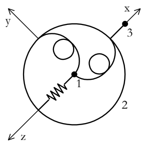
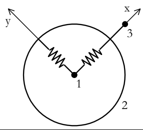
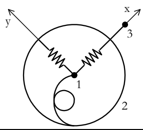
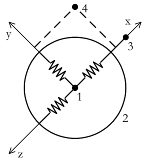
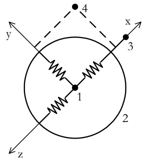
Name	Geometry	Title	Freedoms	Product Version
TTS3		THICK SHELL flat element in 3D	U, V, W, qa, qbor U, V, W, qx, qy, qz	Standard
TTS6		THICK SHELL curved element in 3D	U, V, W, qa, qbor U, V, W, qx, qy, qz	Plus
QTS4		THICK SHELL flat element in 3D	U, V, W, qa, qbor U, V, W, qx, qy, qz	Standard
QTS8	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

Membrane Elements

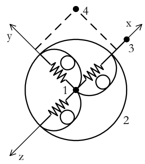
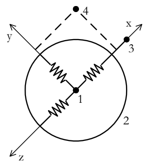
Name	Geometry	Title	Freedoms	Product Version
BXM2		AXISYMMETRIC membrane element in 2D	U, V	Standard
BXM3		AXISYMMETRIC membrane element in 2D	U, V	Standard
TSM3		SPACE membrane element in 3D	U, V, W	Standard

SMI4		SPACE membrane element in 3D	U, V, W	Standard

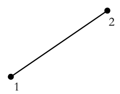
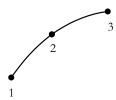
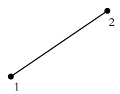
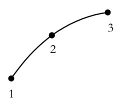
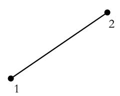
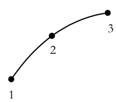
Joint Elements

Name	Geometry	Title	Freedoms	Product Version
JNT3		JOINT ELEMENT in 2D for bars, plane stress and plane strain	U, V	Standard
JPH3		JOINT ELEMENT in 2D for engineering and Kirchhoff beams	U, V, qz	Standard
JF3		JOINT ELEMENT in 2D for grillage beams and plates	W, qx, qy	Standard
JAX3		JOINT ELEMENT in 2D for axisymmetric solids	U, V	Standard
JXS3		JOINT ELEMENT in 2D for axisymmetric shells	U, V, qz	Standard
JNT4		JOINT ELEMENT in 3D for bars, solids and space membranes	U, V, W	Standard
JL43		JOINT ELEMENT in 3D for corner nodes of semiloof elements	U, V, W	Standard

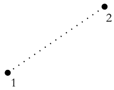
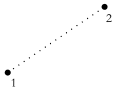
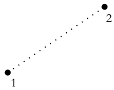
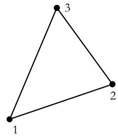
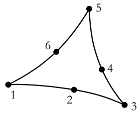
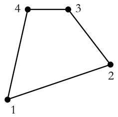
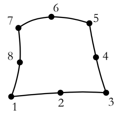
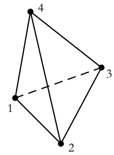
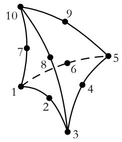
Joint Elements

Name	Geometry	Title	Freedoms	Product Version
JSH4 JL46		JOINT ELEMENT in 3D for engineering and Kirchhoff beams and the end/corner nodes of semiloof elements	U, V, W, qx, qy, qz	Standard
JSL4		JOINT ELEMENT in 3D for mid-side nodes of semiloof elements	U, V, W, q1, q2	Plus

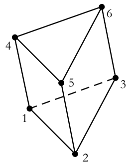
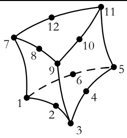
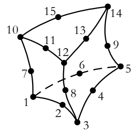
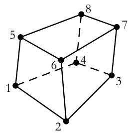
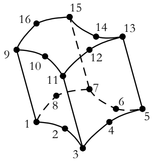
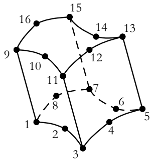
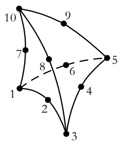
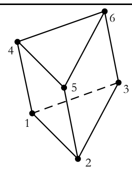
Thermal / Field Elements

Name	Geometry	Title	Freedoms	Product Version
BFD2		THERMAL BAR element in 2D	F	Standard
BFD3		THERMAL BAR element in 2D	F	Standard
BFX2		Axisymmetric THERMAL MEMBRANE element in 2D	F	Standard
BFX3		Axisymmetric THERMAL MEMBRANE element in 2D	F	Standard
BFS2		THERMAL BAR element in 3D	F	Standard
BFS3		THERMAL BAR element in 3D	F	Standard

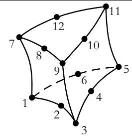
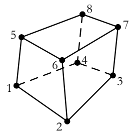
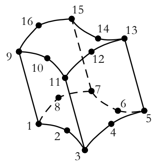
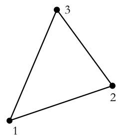
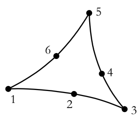
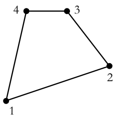
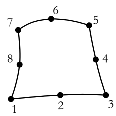
Thermal / Field Elements

Name	Geometry	Title	Freedoms	Product Version
LFD2		THERMAL LINK element in 2D	F	Standard
LFX2		Axisymmetric THERMAL LINK element in 2D	F	Standard
LFS2		THERMAL LINK element in 3D	F	Standard
TFD3		PLANE FIELD element in 2D	F	Standard
TFD6		PLANE FIELD element in 2D	F	Standard
QFD4		PLANE FIELD element in 2D	F	Standard
QFD8		PLANE FIELD element in 2D	F	Standard
TF4		SOLID FIELD element in 3D	F	Standard
TF10		SOLID FIELD element in 3D	F	Plus

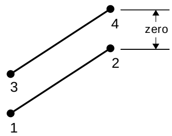
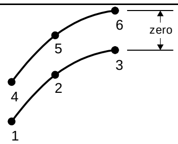
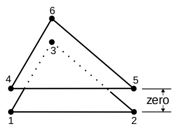
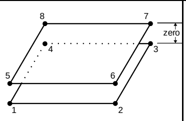
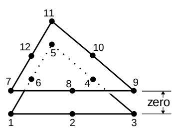
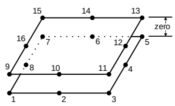
Thermal / Field Elements

Name	Geometry	Title	Freedoms	Product Version
PF6		SOLID FIELD element in 3D	F	Standard
PF12		SOLID FIELD element in 3D	F	Plus
PF15		SOLID FIELD element in 3D	F	Plus
HF8		SOLID FIELD element in 3D	F	Standard
HF16		SOLID FIELD element in 3D	F	Plus
HF20		SOLID FIELD element in 3D	F	Plus
TF10S		SOLID FIELD composite element in 3D	F	Plus
PF6C		SOLID FIELD composite element in 3D	F	Plus

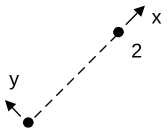
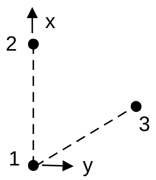
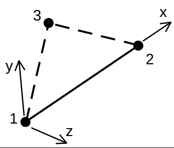
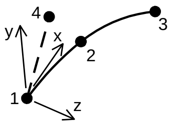
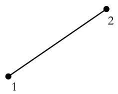
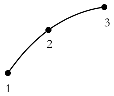
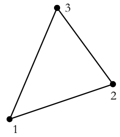
Thermal / Field Elements

Name	Geometry	Title	Freedoms	Product Version
PF12C		SOLID FIELD composite element in 3D	F	Plus
HF8C		SOLID FIELD composite element in 3D	F	Plus
HF16C		SOLID FIELD composite element in 3D	F	Plus
TXF3		AXISYMMETRIC FIELD element in 2D	F	Standard
TXF6		AXISYMMETRIC FIELD element in 2D	F	Standard
QXF4		AXISYMMETRIC FIELD element in 2D	F	Standard
QXF8		AXISYMMETRIC FIELD element in 2D	F	Standard

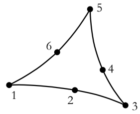
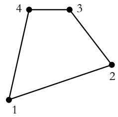
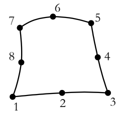
Interface Elements

Name	Geometry	Title	Freedoms	Product Version
IPN4 **IAX4**		INTERFACE ELEMENT in 2D for modelling delamination and axisymmetric crack propagation	U, V	Plus
IPN6 **IAX6**		INTERFACE ELEMENT in 2D for modelling delamination and axisymmetric crack propagation	U, V	Plus
IS6		INTERFACE ELEMENT in 3D for modelling delamination and crack propagation	U, V, W	Plus
IS8		INTERFACE ELEMENT in 3D for modelling delamination and crack propagation	U, V, W	Plus
IS12		INTERFACE ELEMENT in 3D for modelling delamination and crack propagation	U, V, W	Plus
IS16		INTERFACE ELEMENT in 3D for modelling delamination and crack propagation	U, V, W	Plus

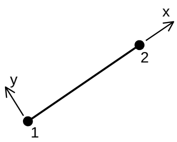
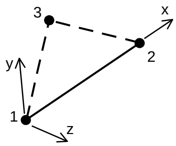
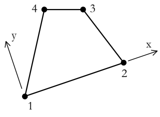
Non-Structural Mass Elements

Name	Geometry	Title	Freedoms	Product Version
PM2		NON-STRUCTURAL MASS ELEMENT in 2D to model mass at a point	U, V	Plus
PM3		NON-STRUCTURAL MASS ELEMENT in 3D to model mass at a point	U, V, W	Plus
LMS3		NON-STRUCTURAL MASS ELEMENT in 3D to model mass along an edge	U, V, W	Plus
LMS4		NON-STRUCTURAL MASS ELEMENT in 3D to model mass along an edge	U, V, W	Plus
LM2		NON-STRUCTURAL MASS ELEMENT in 2D to model mass along an edge	U, V	Plus
LM3		NON-STRUCTURAL MASS ELEMENT in 2D to model mass along an edge	U, V	Plus
TM3		NON-STRUCTURAL MASS ELEMENT in 3D to model mass on a surface.	U,V,W	Plus

Non-Structural Mass Elements

Name	Geometry	Title	Freedoms	Product Version
TM6		NON-STRUCTURAL MASS ELEMENT in 3D to model mass on a surface.	U,V,W	Plus
QM4		NON-STRUCTURAL MASS ELEMENT in 3D to model mass on a surface.	U,V,W	Plus
QM8		NON-STRUCTURAL MASS ELEMENT in 3D to model mass on a surface.	U,V,W	Plus

Rigid Elements

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

		Bars		Beams													
		BAR2, BAR3	BRS2, BRS3	BEAM	BMS3	GRIL	BTS3	BMI21	BMX21	BMI22 BMI31 BMI33	BMX22, BMX31, BMX32	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)															✓	✓
	U, V, W, qx, qy																
	U, V, W, qx, qy, qz				✓		✓	✓	✓	✓	✓						
Material Properties	Linear (Isotropic)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Linear (Orthotropic)																
	Linear (Anisotropic)																
	Linear (Rigidities)						✓	✓		✓		✓		✓	✓	✓	
	Matrix																
	Joint																
	Concrete																
	Stress Resultant						✓	✓		✓		✓		✓		✓	
	Tresca	✓	✓						✓		✓		✓		✓		✓
	Drucker-Prager	✓	✓						✓		✓		✓		✓		✓
	Mohr-Coulomb	✓	✓						✓		✓		✓		✓		✓
	Optimised Implicit Von	✓	✓						✓		✓		✓		✓		✓

Bar and Beam Element Summary

		Bars		Beams													
		BAR2, BAR3	BRS2, BRS3	BEAM	BMS3	GRIL	BTS3	BMI21	BMX21	BMI22 BMI31 BMI33	BMX22, BMX31, BMX32	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	+	+	+	+
	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 (CEB-FIP_90, Eurocode_2, General, User)	✓					✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Rubber																
	Generic Polymer																
	Multi-linear	✓	✓														
	Composite																
	Field																
Loading	Prescribed Value (PDSP,TPDSP)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Types	Concentrated Loads (CL)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Element Load (ELDS)			✓	✓	✓	✓	✓	✓	✓	✓	✓	✓				✓

Bar and Beam Element Summary

		Bars		Beams													
		BAR2, BAR3	BRS2, BRS3	BEAM	BMS3	GRIL	BTS3	BMI21	BMX21	BMI22 BMI31 BMI33	BMX22, BMX31, BMX32	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	+	+	+	+
	Distributed Load (UDL)			✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	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)	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Target Stress/Strain (TSSIE,TSSIA)	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Target Stress/Strain (TSSIG)	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Temperature (TEMP,TMPE)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Field Loads																

Bar and Beam Element Summary

		Bars		Beams													
		BAR2, BAR3	BRS2, BRS3	BEAM	BMS3	GRIL	BTS3	BMI21	BMX21	BMI22 BMI31 BMI33	BMX22, BMX31, BMX32	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	+	+	+	+
	Temperature Dependent Loads																
Nonlinear Geometry	Total Lagrangian	✓	✓					✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Updated Lagrangian											✓	✓	✓	✓		
	Eulerian																
	Co-rotational	✓	✓				✓	✓	✓	✓	✓						
Integration Schemes	Explicitly Integrated			✓	✓	✓											
	Numerically Integrated	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Mass Modelling	Consistent Mass (default)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Lumped Mass	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

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	U, V	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓	✓	✓	✓	✓		
Freedoms	U, V, W																	✓
(corner)	U, V, (P)										✓						✓	
Material	Linear (Isotropic)	✓	✓	✓	✓	✓	✓		✓	✓	✓	✓	✓		✓	✓	✓	✓
Properties	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)	✓	✓	✓		✓	✓		✓			✓	✓		✓	✓		
	Creep (Eurocode)	✓	✓	✓		✓	✓		✓			✓	✓		✓	✓		
	Damage (Simo, Oliver)	✓	✓	✓		✓	✓		✓	✓	✓	✓	✓		✓	✓	✓	
	Viscoelastic					✓	✓			✓	✓	✓	✓		✓	✓	✓	

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	+	+	+
	Shrinkage (CEB-FIP, Eurocode. General, User)	✓		✓		✓	✓		✓		✓	✓	✓	✓	✓		✓	
	Rubber (Ogden, Mooney-Rivlen, Neo-Hookean, Hencky)		✓				✓	✓						✓				
	Generic Polymer		✓	✓		✓	✓		✓		✓	✓	✓		✓		✓	
	Composite																	
	Field																	
Loading Types	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)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	
	Target Stress/Strain (TSSIE,TSSIA)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Target Stress/Strain (TSSIG)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓	✓	✓
	Temperature (TEMP,TMPE)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Field Load																	
	Temp Dependent Load																	

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	+	+	+
Nonlinear Geometry	Total Lagrangian	✓	✓	✓		✓	✓		✓		✓	✓	✓		✓		✓	
	Updated Lagrangian	✓	✓	✓		✓	✓		✓		✓	✓	✓		✓		✓	
	Eulerian	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	
	Co-rotational	✓	✓	✓		✓	✓		✓		✓							
Integration Schemes	Explicitly Integrated																	
	Numerically Integrated	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Mass Modelling	Consistent Mass (default)	✓	✓	✓		✓	✓	✓	✓		✓	✓	✓	✓	✓		✓	✓
	Lumped Mass	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

* Supported in LUSAS Solver but not supported in LUSAS Modeller.

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)	U, V													
	U, V, W	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	
	U, V, W (P)													✓
Material Properties	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)	✓	✓	✓		✓	✓		✓			✓	✓	
	Damage	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Viscoelastic	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓

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	+	+	+	+	+	+
	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													
Loading Types	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 (SSR,SSRE)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Residual Stress (SSRG)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

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	+	+	+	+	+	+
	Target Stress/Strain (TSSIE,TSSIA)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Target Stress/Strain (TSSIG)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓
	Temperature (TEMP,TMPE)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Field Load													
	Temp Dependent Load													
Nonlinear Geometry	Total Lagrangian	✓	✓	✓	✓	✓	✓	✓	✓					✓
	Updated Lagrangian	✓	✓	✓	✓	✓	✓	✓	✓					✓
	Eulerian	✓	✓	✓	✓	✓	✓	✓	✓				✓	✓
	Co-rotational	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓
Integration Schemes	Explicitly Integrated													
	Numerically Integrated	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Mass Modelling	Consistent Mass (default)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓
	Lumped Mass	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

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,)						✓							
	U, V, W, qa, qb (U, V, W, qx, qy, qz)								✓	✓	✓	✓		
Material Properties	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)				✓		✓	✓	✓	✓	✓	✓	✓	
	Creep (CEB_FIP_90)				✓			✓	✓	✓	✓	✓		
	Creep (Chinese)				✓			✓	✓	✓	✓	✓		
	Creep (Eurocode)				✓			✓	✓	✓	✓	✓		
	Damage				✓		✓	✓	✓	✓	✓	✓	✓	
	Viscoelastic													
	Shrinkage (CEB-FIP_90, Eurocode_2, General, User)				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

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
	Rubber (Ogden, Mooney-Rivlin, Neo-Hookean, Hencky)												✓	
	Generic Polymer													
	Composite (Composite Shell)						·	✓	✓	✓	✓	✓		
	Field													
Loading Types	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)				✓		✓	✓	✓	✓	✓	✓	✓	
	Target Stress/Strain (TSSIE,TSSIA)	✓	✓	✓	✓	✓	✓						✓	✓
	Target Stress/Strain (TSSIG)				✓		✓	✓	✓	✓	✓	✓	✓	✓
	Temperature (TEMP,TMPE)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Field Load													
	Temp Dependent Loads													
Nonlinear Geometry	Total Lagrangian				✓		✓	✓	✓	✓	✓	✓	✓	
	Updated Lagrangian				✓		✓	✓						
	Eulerian													
	Co-rotational						✓							
Integration Schemes	Explicitly Integrated													
	Numerically Integrated	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Mass Modelling	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 Types	Prescribed value (PDSP,TPDSP)	✓	✓	✓	✓	✓	✓	✓	✓
	Concentrated Load (CL)	✓	✓	✓	✓	✓	✓	✓	✓
	Element Load								
	Distributed Load								

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**	**+**
	Body Force(CBF)	✓	✓	✓	✓	✓	✓	✓	✓
	Body Force (BFP,BFPE)								
	Velocities (VELO)	✓	✓	✓	✓	✓	✓	✓	✓
	Acceleration (ACCE)	✓	✓	✓	✓	✓	✓	✓	✓
	Initial Stress/Strain (SSI,SSIE)	✓	✓	✓	✓	✓	✓	✓	✓
	Initial Stress/Strain (SSIG)								
	Residual Stress								
	Target Stress/Strain (TSSIE,TSSIA)	✓	✓	✓	✓	✓	✓	✓	✓
	Target Stress/Strain (TSSIG)								
	Temperature (TEMP,TMPE)	✓	✓	✓	✓	✓	✓	✓	✓
	Field Load								
	Temp Dependent Load								
Nonlinear Geometry	Total Lagrangian								
	Updated Lagrangian								
	Eulerian								
	Co-rotational								
Integration Schemes	Explicitly Integrated								
	Numerically Integrated	✓	✓	✓	✓	✓	✓	✓	✓
Mass Modelling	Consistent Mass (default)								
	Lumped Mass	✓	✓	✓	✓	✓	✓	✓	✓

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

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
Product Version	**LT, Standard (S) or Plus (+)**	S	S	S	S	S	S	S	S	S	S	S	S	S	+	+	+	+	S	S
Freedoms	F	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Material Properties	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 (Arbitary Conv/Rad)				✓	✓	✓													
Loading Types	Prescribed Value (PDSP,TPDSP)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Concentrated Loads (CL)	✓	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Element Load																			
	Distributed Load (UDL)															✓	✓	✓	✓	✓
	Distributed Load (FLD)	✓	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Body Force (CBF,	✓	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

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
Product Version	**LT, Standard (S) or Plus (+)**	S	S	S	S	S	S	S	S	S	S	S	S	S	+	+	+	+	S	S
	BFP, BFPE)																			
	Velocity																			
	Acceleration																			
	Initial Stress/Strain																			
	Residual Stress																			
	Temperature (TEMP,TMPE)																			
	Field Load																			
	Temperature (TEMP, TMPE)	✓	✓	✓																
	Field Load (ENVT)	✓	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Temp Dep Load (TDET/RIHG)	✓	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Nonlinear Geometry	Total Lagrangian																			
	Updated Lagrangian																			
	Eulerian																			
	Co-rotational																			
Integration Schemes	Explicitly Integrated																			
	Numerically Integrated	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Specific Heat	Consistent (default)	✓	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
	Lumped	✓	✓	✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

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											
	U, V, W, qx, qy, qz											
	U, V, W, q1, q2											
Material Properties	Linear										✓	✓
	Matrix											
	Joint											
	Mass					✓	✓	✓	✓	✓		
	Concrete											
	Elasto-Plastic											
	Creep											
	Damage											
	Shrinkage											
	Interface	✓	✓	✓	✓							
	Rubber											
	Generic Polymer											
	Stress Potential											
	Composite											
	Field											
Loading Types	Prescribed value (PDSP,TPDSP)	✓	✓	✓	✓						✓	✓
	Concentrated Loads (CL)	✓	✓	✓	✓							
	Element Load											
	Distributed Load											
	Body Force (CBF)					✓	✓	✓	✓	✓		
	Body Force (BFP,BFPE)											
	Velocity (VELO)	✓	✓	✓	✓						✓	✓
	Acceleration (ACCE)	✓	✓	✓	✓						✓	✓
	Initial Stress/Strain (SSI,SSIE)											

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 (+)**	+	+	+	+	+	+	+	+	+	+	+
	Initial Stress/Strain (SSIG)											
	Residual Stress											
	Target Stress/Strain (TSSIE,TSSIA)											
	Target Stress/Strain (TSSIG)											
	Temperature (TEMP,TMPE)	✓	✓	✓	✓							
	Field Load											
	Temp Dependent Load											
Nonlinear Geometry	Total Lagrangian										✓*	✓*
	Updated Lagrangian										✓*	✓*
	Eulerian										✓*	✓*
	Co-rotational	✓		✓							✓*	✓*
Integration Schemes	Explicitly Integrated											
	Numerically Integrated	✓	✓	✓	✓	✓	✓	✓	✓	✓		
Mass Modelling	Consistent Mass (default)					✓	✓	✓	✓	✓		
	Lumped Mass					✓	✓	✓	✓	✓		

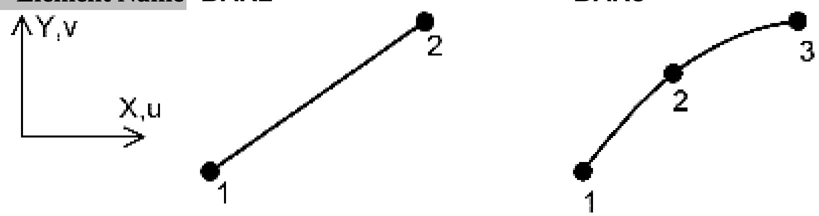
* Dependent upon the other surface (deformable surface) that the element is in contact with.

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.
Freedoms	U, V at each node
Node Coordinates	X, Y at each node

Geometric Properties

A_1 ... 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, Ωx, Ωy, Ωz, αz

	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. Fx, εx, σx , εx
	SSIG	Initial stresses/strains at Gauss points. F , εx, σx , εx
Residual Stresses	SSR, SSRE	Not applicable.
	SSRG	Residual stresses at Gauss points. Components (nonlinear material models): 0, 0, σx
Target Stress/Strains	TSSI, TSSIA	Target stresses/strains at nodes/for element. Fx, εx, σx , εx
	TSSIG	Target stresses/strains at nodes/for element. F, εx, σx , εx
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: εx
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	For large displacements and small strains.

Integration Schemes

Stiffness	Default.	1-point (BAR2), 2-point (BAR3).
	Fine (see *Options*).	2-point (BAR2).
Mass	Default.	2-point (BAR2), 3-point (BAR3).
	Fine (see *Options*).	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.
229	Co-rotational geometric nonlinearity.

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

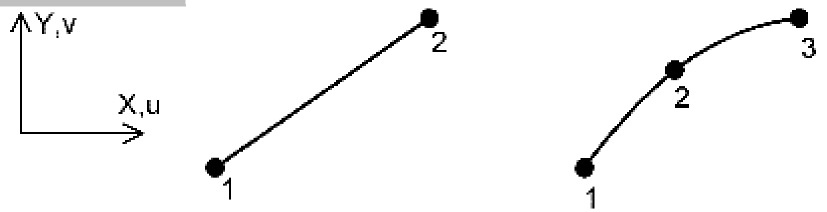
Theory

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.
Freedoms	U, V, W at each node
Node Coordinates	X, Y, Z at each node.

Geometric Properties

A_1 ... 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, Ωx, Ωy, Ωz, αx, αy, αz
	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. Fx, εx, σx , εx
	SSIG	Initial stresses/strains at Gauss points. F , εx, σx , εx
Residual Stresses	SSR, SSRE	Not applicable
	SSRG	Residual stresses at Gauss points. Components (nonlinear material models): 0, 0,

		σx
Target Stress/Strains	TSSI, TSSIA	Target stresses/strains at nodes/for element. Fx, εx, σx , εx
	TSSIG	Target stresses/strains at nodes/for element. F, εx, σx , εx
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: εx
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	For large displacements and small strains.

Integration Schemes

Stiffness	Default.	1-point (BRS2), 2-point (BRS3).
	Fine (see *Options*).	2-point (BRS2).
Mass	Default.	2-point (BRS2), 3-point (BRS3).
	Fine (see *Options*).	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.
229	Co-rotational geometric nonlinearity.

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

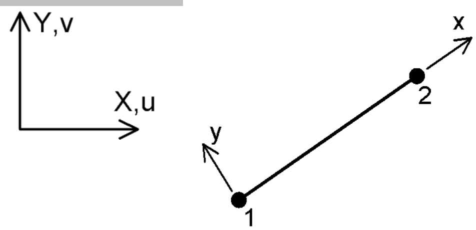
- The 2-node elements are the most effective bar elements for modelling 'stand-alone-elements' such as members of trusses or bars connecting two discrete structures.
- They can be used to model cables in cable-stayed structures.
- Both the 2-noded and 3-noded elements are suitable for modelling reinforcement 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.
Freedoms	U, V, θz: at each node.
Node Coordinates	The element node numbers should be followed by: R restrained (default), F free defined in the order θz at node 1 and then θz at node 2 related to local element axes.

Geometric Properties

A, Izz, Asy, ey	for element
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, θz: 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. Xcbf, Ycbf, Ωx, Ωy, Ωz, αz
	BFP, BFPE	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.	
Target Stress/Strains	Not applicable.	
Temperatures	TEMP, TMPE	Temperatures at nodes/for elements. T, 0, dT/dy, 0, To, 0, dTo/dy, 0
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Force (default):Fx, Fy, Mz: 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 appled 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. When a step by step dynamic analysis is carried out using BEAM elements with distributed loading, the "free body force diagrams" pertaining to applied loading, are not superimposed on the nodal values, to do so would lead to erroneous results until a steady state is reached. It should therefore be noted that different force diagrams will be obtained for BEAM elements if static and dynamic analyses are directly compared
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.

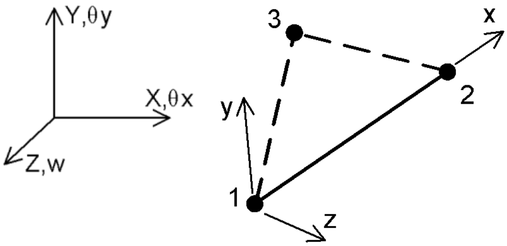
Recommendations on Use

- 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.
Freedoms	U, V, W, θx, θy, θz: at end nodes.
End Releases	The element node numbers should be followed by: R restrained (default), F free defined in the order θy, θz at node 1 and then θy θz at node 2 related to local element axes
Node Coordinates	X, Y, Z: at each node.

Geometric Properties

A, Iyy, Izz, Jxx, Asz, Asy, ez, ey	for element
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:	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, W, θx, θy, θz at nodes. Concentrated loads in global directions.
Concentrated Loads	CL	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 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 (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.
	FLD	Not applicable.
Body Forces	CBF	Constant body forces for Element. Xcbf, Ycbf, Zcbf, Ωx, Ωy, Ωz, αx, αy, αz
	BFP, BFPE	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.	
Target Stress/Strains	Not applicable.	
Temperatures	TEMP, TMPE	Temperatures at nodes/for elements. 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, Fy, Fz, Mx, My, Mz: 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 appled 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. When a step by step dynamic analysis is carried out using BMS3 elements with distributed loading, the "free body force diagrams" pertaining to applied loading, are not superimposed on the nodal values, to do so would lead to erroneous results until a steady state is reached. It should therefore be noted that different force diagrams will be obtained for BMS3 elements if static and dynamic analyses are directly compared.
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 Iyz of the section must equal zero.

Recommendations on Use

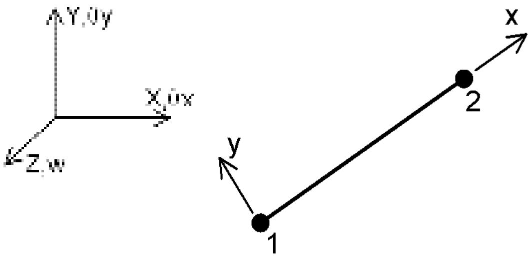
- The element can be used to model 3D frame structures, or act as a stiffener for the **QTS4, TTS3** and **QSI4, TS3** flat shell elements. Linear, eigen, and dynamic analysis procedures are fully supported with this element.
- The element can be used to model cables in cable-stayed structures.
- The 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 θy at node 1 and then θy at node 2 related to local element axes
Freedoms	W, θx, θy: at each node.
Node Coordinates	X, Y: at each node.

Geometric Properties

A, Iyy, Izz, Jxx, Asz, EFW	for element
A	Cross sectional area
Iyy, Izz	2nd moments of area about local y, z axes (see **Definition** and *Notes*)
Jxx	**Torsional constant**
Asz	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, θx, θy: 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

		and global Z are coincident).
	FLD	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.	
Target Stress/Strains	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 *Notes*). 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, (Izz), is only required when assembling the mass matrix.
5. Strains are not available for GRIL elements.
6. When a step by step dynamic analysis is carried out using BEAM elements with distributed loading, the "free body force diagrams" pertaining to applied loading, are not superimposed on the nodal values, to do so would lead to erroneous results until a steady state is reached. It should therefore be noted that different force diagrams will be obtained for BEAM elements if static and dynamic analyses are directly compared.
7. Though this element cannot model nonlinear behaviour it can be mixed with other elements in a nonlinear analysis.
8. For restrictions on the use of **Wood-Armer** with grillages refer to the LUSAS User Guide and Theory Manual.
9. 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.
10. 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.

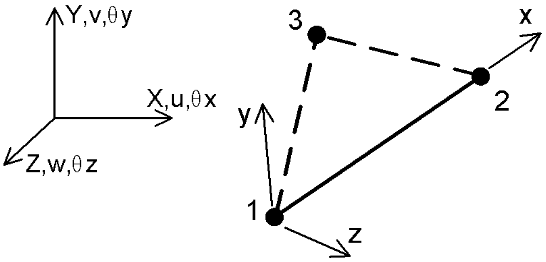
Recommendations on Use

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.
Freedoms	U, V, W, θx, θy, θz: 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, θx, θy, θz at node 1 and then U, V, W, θx, θy, θz at node 2 related to local element axes, see *Notes*.)
Node Coordinates	X, Y, Z: at each node.

Geometric Properties

A, Iyy, Izz, Jxx, Asz, Asy, Iy, Iz, Iyz, ez, ey	for element.
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 *Notes*)
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, θx, θy, θz: at nodes.
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 *Notes*)
	FLD	Not applicable.
Body Forces	CBF	Constant body forces for Element. Xcbf, Ycbf, Zcbf, Ωx, Ωy, Ωz, αx, αy, αz
	BFP, BFPE	Not applicable.
Velocities	VELO	Velocities. Vx, Vy, Vz: at nodes.

Accelerations	ACCE	Acceleration. Ax, Ay, Az: at nodes
Initial Stress/Strains	SSI, SSIE, SSIG	Initial stresses/strains at nodes/for element. Fx, Fy, Fz, Mx, My, Mz: axial force, shear forces, torque and moments in local directions. εx, εy, εz, ψx, ψy, ψz: axial, shear and flexural strains in local directions.
Target Stress/Strains	TSSIE, TSSIA, TSSIG	Target stresses/strains at nodes/for element. Fx, Fy, Fz, Mx, My, Mz: axial force, shear forces, torque and moments in local directions. εx, εy, εz, ψx, ψy, ψz: 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: εx, εy, εz, ψx, ψy, ψz: 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, K_T, Asz, Asy, ez, ey (8 parameters), or A, Iyy, Izz, K_T, 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, Zyy^p, Zzz^p, Zy^p, Zz^p, S^p
    - A^p Plastic area (=elastic area)
    - Zyy^p, Zzz^p Plastic moduli for bending about y, z axes
    - Zy^p, Zz^p Plastic moduli for torsion about y, z axes.
    - S^p Plastic area for shear (S^p=0).
    - Where the fully plastic torsional moment = $\sigma y\ (Zy^p + Zz^p)$.

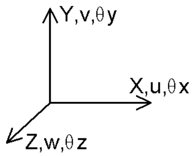
- Note that if eccentricity has been specified the plastic properties must be defined with reference to the **nodal line** and **not** the beam axes, i.e. the eccentricity is not used to automatically modify the plastic properties, they must be defined via modified geometry.

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.

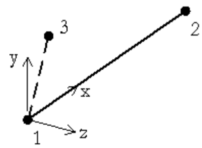
3D Thick Beam Elements

General

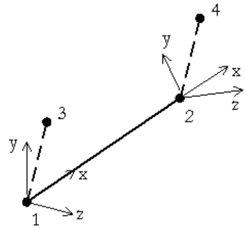
Element Name



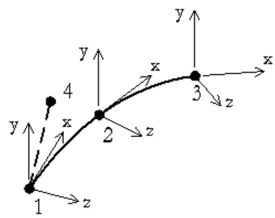
BMI21



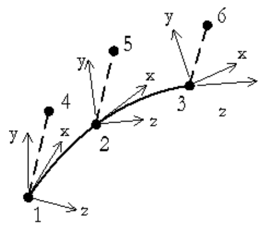
BMI22



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. The orientation node(s) (3rd node of BMI21, 3rd and 4th nodes of BMI22, 4th node of BMI31, 4th, 5th and 6th nodes of BMI33) are used to define the local xy-plane.
Freedoms	U, V, W, θx, θy, θz: 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, θx, θy, θz at node 1 and then U, V, W, θx, θy, θz 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, Asz, Asy, Iyz, ez, ey	At each node
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, θx, θy, θz: 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, Ωx, Ωy, Ωz, αx, αy, αz
	BFP, BFPE	Body force potentials at nodes/for element. $\varphi 1$, $\varphi 2$, $\varphi 3$, 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. εx, εy, εz, ψx, ψy, ψz: 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.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. Fx, Fy, Fz, Mx, My, Mz: axial force, shear forces, torque and moments in local directions. εx, εy, εz, ψx, ψy, ψz: axial, shear and flexural strains in local directions.
	TSSIG	Target stresses/strains at Gauss points. These stresses/strains are specified in the same manner as TSSIE and TSSIA.
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: εx, εy, εz, ψx, ψy, ψz: 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	For large displacements and large rotations
Updated Lagrangian	Not applicable.
Eulerian	Not applicable.
Co-rotational	For large displacements and large rotations

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.
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.
229 Co-rotational geometric nonlinearity.
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, Zyy^p, Zzz^p, Zy^p, Zz^p, S^p at each node
    - A^p Plastic area (=elastic area)
    - Zyy^p, Zzz^p Plastic moduli for bending about y, z axes
    - Zy^p, Zz^p Plastic moduli for torsion about y, z axes.
    - S^p Plastic area for shear (S^p=0).

    Where the fully plastic torsional moment = $\sigma y\ (Zy^p + Zz^p)$.
    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. When a step by step dynamic analysis is carried out using BMI elements with distributed loading, the "free body force diagrams" pertaining to applied loading, are not superimposed on the nodal values, to do so would lead to erroneous results until a steady state is reached. It should therefore be noted that different force diagrams will be obtained for BMI elements if static and dynamic analyses are directly compared.
13. OPTION 87 considers large displacements and large rotations using a Total Lagrangian formulation; OPTION 229 considers large displacements and large rotations using a co-rotational formulation. In general the co-rotational formulation works better. When options 87 and 229 are true, a local Total Lagrangian formulation will be used together with a global co-rotational for mulation.

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

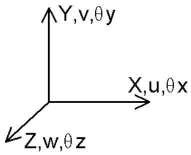
- The elements may be used for linear and material nonlinear analysis of three dimensional beam, frame and arch structures, and can also be used to model cables in cable stayed structures. BMI21 and BMI22 may also be used as a stiffener for the QTS4 shell element; while BMI31 and BMI33 may be used as a stiffener for the QTS8 shell element, e.g. space frames.

- When Solver options 403 and 404 are turned on (as they are by default) the BMI21 element behaves the same as the 2-noded straight beam element BMS3 for linear analysis of structures containing straight members of constant cross-section.

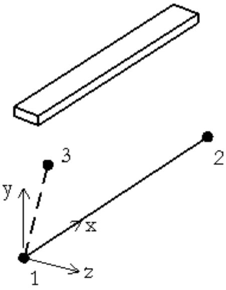
3D Thick Beam Elements with Quadrilateral Cross-Section

General

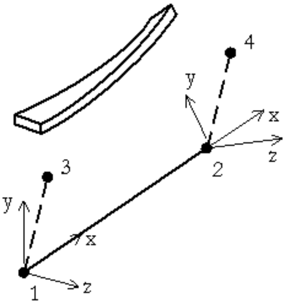
Element Name



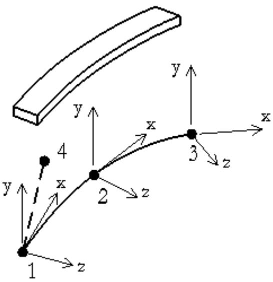
BMX21



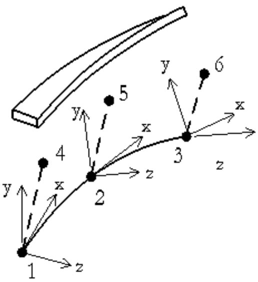
BMX22



BMX31



BMX33



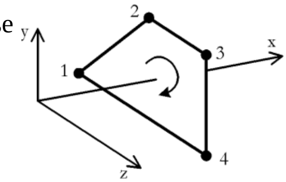
Element Group	Beams
Element Subgroup	Thick Beams
Element Description	Straight and curved isoparametric degenerate thick beam elements in 3D for which shearing deformations are included. The element has a quadrilateral cross section which may vary along the element length. BMX22 and BMX33 can consider initial twist.
Number Of Nodes	3 (BMX21), 4 (BMX22 and BMX31) and 6 (BMX33) with end release conditions. The orientation node(s) (3rd node of BMX21, 3rd and 4th nodes of BMX22, 4th node of BMX31, 4th, 5th and 6th nodes of BMX33) are used to define the local xy-plane.
Freedoms	U, V, W, θx, θy, θz: 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, θx, θy, θz at node 1 and then U, V, W, θx, θy, θz at node 2 and node 3 (only for

	BMX31 and BMX33) related to local element axes (see Notes).
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 nt_{12}, nt_{14}: 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 Polymer	Not applicable	
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, W, θx, θy, θz: at active nodes.
Concentrated Loads	CL	Concentrated loads in global directions. Px, Py, Pz, Mx, My, Mz: at active 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 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, Ωx, Ωy, Ωz, αx, αy, αz
	BFP, BFPE	Body force potentials at nodes/for element. $\varphi 1$, $\varphi 2$, $\varphi 3$, 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	Target stresses/strains at nodes/for element. Components: 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, σx, 0, 0,εx, 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, σx, 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.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. Components: 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, σx, 0, 0,εx, 0, 0) Bracketed terms repeated for each fibre integration point.
	TSSIG	Target stresses/strains at Gauss points. These stresses/strains are specified in the same manner as TSSIE and TSSIA.

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.

Continuum stresses (OPTION 172): σx, σxy, σxz: in local directions.

Strain: εx, εy, εz, ψx, ψy, ψz: Axial, shear, torsional and flexural strains in local directions.

Continuum strains (OPTION 172): εx, εxy, εxz: 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.

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.

172 Form the **rigidity matrix** by numerical cross section integration.

229 Co-rotational geometric nonlinearity.

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, Jxx = Iyy + Izz.
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.
11. OPTION 87 considers large displacements and large rotations using a Total Lagrangian formulation; OPTION 229 considers large displacements and large rotations using a co-rotational formulation. In general the co-rotational formulation works better. When options 87 and 229 are true, a local Total Lagrangian formulation will be used together with a global co-rotational formulation.

Restrictions

- ❑ Ensure mid-side node centrality
- ❑ Avoid excessive element curvature

- ❑ BMX22 and BMX33 are not available for selection currently within LUSAS Modeller.

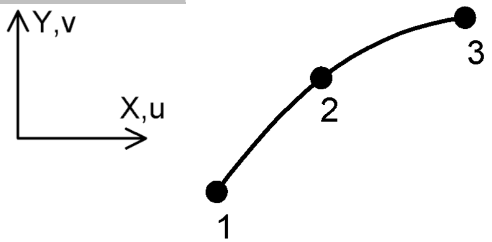
Recommendations on Use

- 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 Name	**BM3**



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
Freedoms	U, V, θz: 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 $Izz=e^2A+I_{na}$ and Iz=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 *Notes*)
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, θz: 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, Ωx, Ωy, Ωz, αz
	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 Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. Fx, Mz, 0: forces, moments in local directions. εx, ψz, 0: strains in local directions.
	SSIG	Initial stresses/strains at Gauss points Fx, Mz, 0: forces, moments in local directions. εx, ψz, 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
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. Fx, Mz, 0: forces, moments in local directions. εx, ψz, 0: strains in local directions.
	TSSIG	Target stresses/strains at Gauss points Fx, Mz, 0: forces, moments in local directions. εx, ψz, 0: strains in local directions.
Field Loads	Not applicable.	

Temp Dependent Loads	Not applicable.

LUSAS Output

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

Strain: εx, εy, ψz: 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, Zzz^p, S^p at each node (i.e. nodes 1, 2, 3)
   - A^p Plastic area (=elastic area)
   - Zzz^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

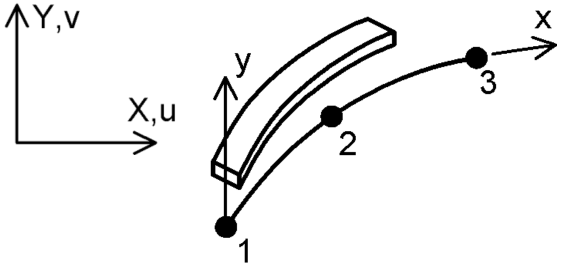
Recommendations on Use

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**

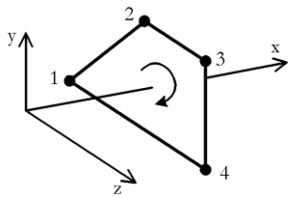


Element Group	Beams
Element Subgroup	**Kirchhoff** Beams
Element Description	Parabolically curved thin beam elements in which shear deformations are excluded. The quadrilateral cross-section may be eccentric and can vary along the element length.
Number Of Nodes	3
Freedoms	U, V, θz: at end nodes. dU: (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 nt_{12}, nt_{14}: 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 *Notes*)
	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	Not applicable	

Polymer Composite	Not applicable

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, θz: 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, Ωx, Ωy, Ωz, αz
	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 Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. Resultants (for linear material models without numerical cross section integration and model 29, see *Notes*): Fx, Mz, 0: forces, moments in local directions. εx, ψz, 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 *Notes*). Fx, Mz, 0: forces, moments in local directions. εx, ψz, 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, (σx, εx). 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): Fx, Mz, 0 (2) Components (all nonlinear material models except 29, also linear material models with numerical cross section integration): 0, 0, 0, 0, 0, 0, (σx) Bracketed term repeated for each fibre integration point.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. Resultants (for linear material models without numerical cross section integration and model 29, see *Notes*): Fx, Mz, 0: forces, moments in local directions. εx, ψz, 0: strains in local directions.
	TSSIG	Target stresses/strains at Gauss points. (1) Resultants (for linear material models without numerical cross section integration and model 29, see *Notes*). Fx, Mz, 0: forces, moments in local directions. εx, ψz, 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, (σx, εx). Bracketed terms repeated at each fibre integration point.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element T, 0, dT/dy, 0, To, 0, dTo/dy, 0: in local directions.
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

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

Continuum stresses (OPTION 172): σx: in local directions.

Strain: εx, ψz, 0 : axial, flexural strains in local directions.

Continuum strains (OPTION 172): εx: 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.
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

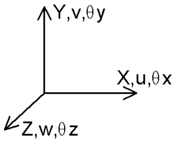
Recommendations on Use

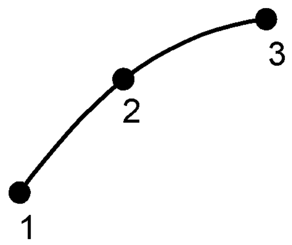
The element may be used for linear and nonlinear analysis of two dimensional beam, frame and arch structures. The 2-noded straight beam (BEAM) is more effective for linear analysis of structures containing straight members of constant cross-section, e.g. plane frames.

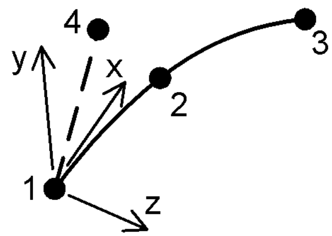
3D Kirchhoff Thin Beam Elements

General

Element Name	BS3	BS4







Element Group	Beams
Element Subgroup	**Kirchhoff** Beams
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.
Freedoms	U, V, W, θx, θy, θz: at end nodes (1 and 3) dU, dθx:(relative displacement/rotation) at mid-length node.
Node Coordinates	X, Y, Z: at each node.

Geometric Properties

A, Iyy, Izz, Jxx, Iy, Iz, Iyz, ez, ey	At each node
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:	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 *Notes*)
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, θx, θy, θz: at end nodes (1 and 3). dU, dθx: 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, Ωx, Ωy, Ωz
	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. εx, ψy, ψz, ψxz, ψxy, 0: axial, flexural and torsional strains in local directions. Total torque = Txz + Txy, total torsional strain = yxz + ψxy.
	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 = yxz + ψxy.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. Fx, My, Mz, Txz, Txy, 0: axial force, moments and torques in local directions. εx, ψy, ψz, ψxz, ψxy, 0: axial, flexural and torsional strains in local directions. Total torque = Txz + Txy, total torsional strain = yxz + ψxy.
	TSSIG	Not applicable.
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 Dependent Loads	Not applicable.	

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: εx, ψy, ψz, ψxz, ψxy, 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 *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.
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 beam 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, Zyy^p, Zzz^p, Zy^p, Zz^p, S^p at each node (i.e. nodes 1, 2, 3).
    - A^p Plastic area (=elastic area)
    - Zyy^p, Zzz^p Plastic moduli for bending about y, z axes
    - Zy^p, Zz^p Plastic moduli for torsion about y, z axes.
    - S^p Plastic area for shear (S^p=0).

    Where the fully plastic torsional moment = $\sigma y\ (Zy^p + Zz^p)$.
    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

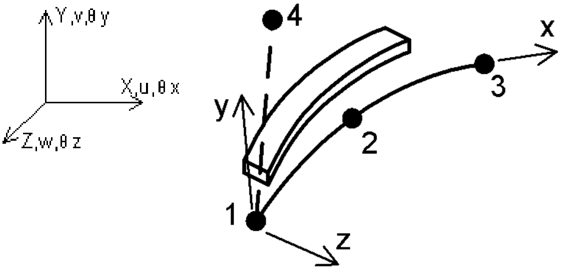
Recommendations on Use

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**

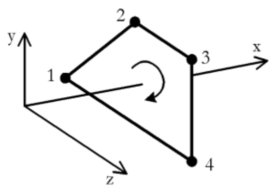


Element Group	Beams
Element Subgroup	Kirchhoff Beams
Element Description	Curved beam elements in 3D for which shearing deformations are excluded. The element has a quadrilateral cross section which may vary along the element length.
Number Of Nodes	4. The 4th node is used to define the local xy-plane.
Freedoms	U, V, W, θx, θy, θz: at the end nodes (1 and 3) dU, dθx: (relative displacement/rotation) at the 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 nt_{12}, nt_{14}: 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	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, θx, θy, θz: at the end nodes. dU, dθx: 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, Ωx, Ωy, Ωz, αx, αy, αz
	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, (σx, σxy, σxz, σyz, εyz, εx, εxz, εyz) 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, (σx, σxy, σxz, σyz,) 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.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. Components: 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, (σx, σxy, σxz, σyz, εyz, εx, εxz, εyz) Bracketed terms repeated for each fibre integration point.
	TSSIG	Target stresses/strains at Gauss points. These stresses/strains are specified in the same manner as TSSIE and TSSIA.
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): σx, σxy, σxz, σyz: in local directions.

Strain: εx, ψy, ψz, ψxz, ψxy: axial, flexural and torsional strains in local directions.

Continuum strains (OPTION 172): εx, εxy, εxz, εyz: 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 *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.
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, Assumptions and Limitations

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 beam 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 Iyy and Izz, Jxx = Iyy + Izz.
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

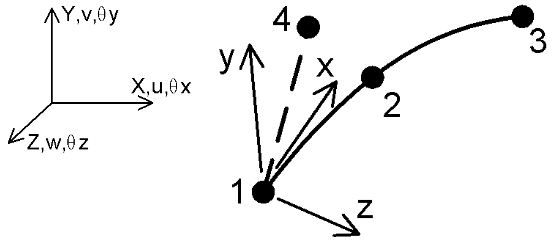
Recommendations on Use

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.
Freedoms	U, V, W, θx, θy, θz: at end nodes (1 and 3). U, V, W, θ_1, θ_2: at mid-side node (node 2) (see *Notes*).
Node Coordinates	X, Y, Z: at each node.

Geometric Properties

A, Iyy, Izz, Jxx, Iy, Iz, Iyz, ez, ey	at nodes 1, 2 and 3
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:	MATERIAL PROPERTIES (Elastic: Isotropic)
	Rigidities:	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 *Notes*)
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, θx, θy, θz: 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, M_1, M_2: at mid-side node (M_1 and M_2 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: force/unit length in local directions for element.
	FLD	Not applicable.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf, Ωx, Ωy, Ωz, αx, αy, αz
	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. Fx, My, Mz, Txz, Txy, 0 in local directions. εx, ψy, ψz, ψxz, ψxy, 0: in local directions. (see *Notes*). Total torque = Txz + Txy
	SSIG	Not applicable.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. Resultants (nonlinear model 29): Fx, My, Mz, Txz, Txy, 0: in local directions.

	SSRG	Not applicable.
Target Stress/Strains	TSSE, TSSIA	Target stresses/strains at nodes/for element. Fx, My, Mz, Txz, Txy, 0 in local directions. εx, ψy, ψz, ψxz, ψxy, 0: in local directions. (see *Notes*). Total torque = Txz + Txy
	TSSIG	Not applicable.
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: in local directions. (Total torque = Txz + Txy)

Strain: εx, ψy, ψz, ψxz, ψxy: in local directions. (see *Notes*). Total torsional strain = ψxz + ψxy
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, Zyy^p, Zzz^p, Zy^p, Zz^p, S^p at each node (i.e. nodes 1, 2, 3).
   - A^p Plastic area (=elastic area)
   - Zyy^p, Zzz^p Plastic moduli for bending about y, z axes
   - Zy^p, Zz^p Plastic moduli for torsion about y, z axes.
   - S^p Plastic area for shear (S^p=0).

   Where the fully plastic torsional moment = $\sigma y\ (Zy^p + Zz^p)$

6. 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

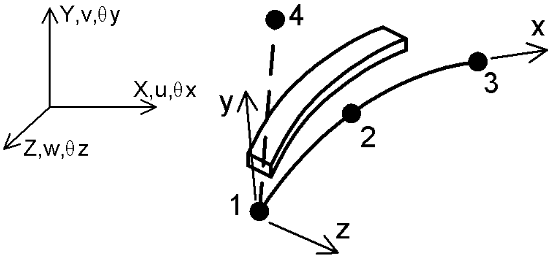
- The primary use of this element is to provide a beam stiffener for the semiloof shell (QSL8) for analysing stiffened shell structures.

- 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**

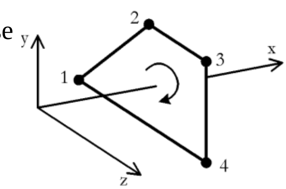


Element Group	Beams
Element Subgroup	Semiloof Beams
Element Description	A curved beam element in 3D which can be mixed with the semiloof shell element. The element has a quadrilateral cross section which may vary along the element. Shearing deformations are excluded.
Number Of Nodes	4. The 4th node is used to define the local xy-plane.
Freedoms	U, V, W, θx, θy, θz: 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 nt_{12}, nt_{14}: 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, θx, θy, θz: 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, M_1, M_2: at mid-side node (M_1 and M_2 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, Ωx, Ωy, Ωz, αx, αy, αz
	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, (σx, σxy, σxz, σyz, εx, εxy, εxz, εyz) 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, (σx, σxy, σxz, σyz,) 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.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. Components: 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, (σx, σxy, σxz, σyz, εx, εxy, εxz, εyz) Bracketed terms repeated for each fibre integration point.
	TSSIG	Target stresses/strains at Gauss points. These stresses/strains are specified in the same manner as TSSIE and TSSIA
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 Dependent Loads	Not applicable.	

LUSAS Output

Solver Force (default): Fx, My, Mz, Txz, Txy, Fy, Fz: in local directions. Total torque = Txz+Txy.

Continuum stresses (Option 172): σx, σxy, σxz, σyz: in local directions.

Strain/curvatures (default): εx, ψy, ψz, ψxz, ψxy, γyz: in local directions (see *Notes*). Total torsional strain = ψxy + ψyz.

Continuum strains (Option 172): εx, εxy, εxz, εyz: 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

4. 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.
5. 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.
6. The torsional constant is estimated from the computed values for Iyy and Izz, Jxx = Iyy + Izz.
7. 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.
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.
10. 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

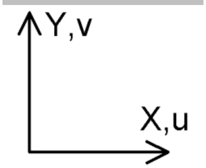
- The element's primary use is to provide a beam stiffener for the semiloof shell (QSL8) for analysing stiffened shell structures.
- The BSX4 element is more effective for linear analysis of 3D frame structures with curved members and nonlinear analysis of three dimensional beam, frame and arch structures.
- 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.

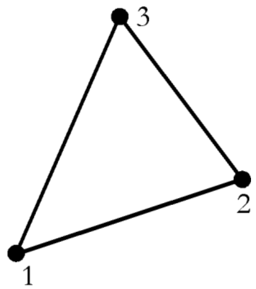
Chapter 3 : 2D Continuum Elements.

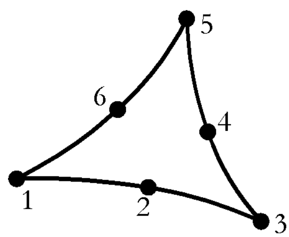
2D Plane Stress Continuum Elements

General

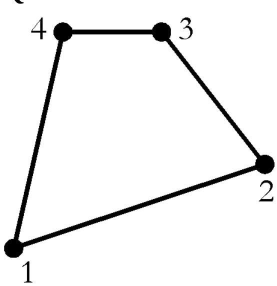
Element Name **TPM3** **TPM6**

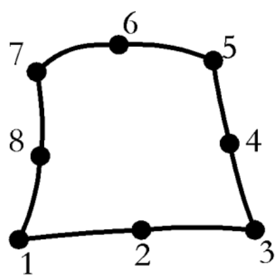






QPM4 **QPM8**





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.
Freedoms	U, V: at each node.
Node Coordinates	X, Y: at each node.

Geometric Properties

$t_1 \ldots t_n$	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

	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, Ωx, Ωy, Ωz, αz
	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. σx, σy, σxy: global stresses. εx, εy, γxy: global strains.
	SSIG	Initial stresses/strains at Gauss points. σx, σy, σxy: global stresses. εx, εy, γxy: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σx, σy, σxy: global stresses.
	SSRG	Residual stresses at Gauss points. σx, σy, σxy: global stresses.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. σx, σy, σxy: global stresses. εx, εy, γxy: global strains.
	TSSIG	Target stresses/strains at Gauss points. σx, σy, σxy: global stresses. εx, εy, γxy: global strains.
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): σx, σy, σxy, σmax, σmin, β, σs, σe (see **description of principal stresses**)

Strain: εx, εy, γxy, εmax, εmin, β, εs, εe

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.
5. If applying an initial stress/strain or thermal load that varies across an element, a higher order element (6 or 8 nodes) should be used. A limitation of the standard isoparametric approach when used for lower order elements (3 or 4 nodes) is that only constant stress/strain fields can be imposed correctly.

Restrictions

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

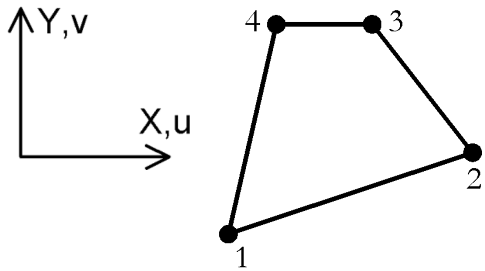
Recommendations on Use

- 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 Subgroup	Plane Stress Continuum
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.
Freedoms	U, V: at each node.
Node Coordinates	X, Y: at each node.

Geometric Properties

$t_1 \ldots t_n$	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	Not applicable	
Shrinkage		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

		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, Ωx, Ωy, Ωz, αz
	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. σx, σy, σxy: global stresses. εx, εy, γxy: global strains.
	SSIG	Initial stresses/strains at Gauss points. σx, σy, σxy: global stresses. εx, εy, γxy: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σx, σy, σxy: global stresses.
	SSRG	Residual stresses at Gauss points. σx, σy, σxy: global stresses.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. σx, σy, σxy: global stresses. εx, εy, γxy: global strains.
	TSSIG	Target stresses/strains at Gauss points. σx, σy, σxy: global stresses. εx, εy, γxy: global strains.
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: Nx, Ny, Nxy, Nmax, Nmin, β, Ns, Ne

Stress (default): σx, σy, σxy, σmax, σmin, β, σs, σe (see **description of principal stresses**)

Strain: εx, εy, γxy, εmax, εmin, β, εs, εe

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 Kirchoff 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.

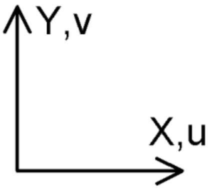
Recommendations on Use

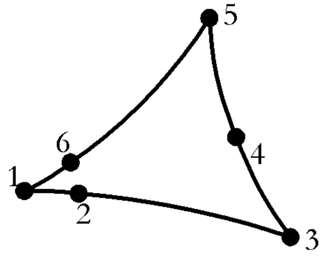
These elements exhibit an improved performance when compared with the parent element QPM4. The integration rules are the same as those given for QPM4, but the elements do not suffer from locking due to parasitic shear when the material approaches the incompressible limit. The elements are also free of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

2D Plane Stress Continuum Crack Tip Elements

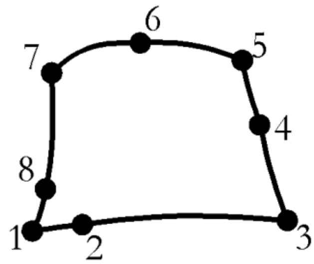
General

Element Name	TPK6	QPK8





Crack specified at Node 1



Crack specified at Node 1

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... 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	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, Ωx, Ωy, Ωz, αz
	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. σx, σy, σxy: global stresses. εx, εy, γxy: global strains.
	SSIG	Initial stresses/strains at Gauss points. σx, σy, σxy: global stresses. εx, εy, γxy: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σx, σy, σxy: global stresses.
	SSRG	Residual stresses at Gauss points. σx, σy, σxy: global stresses.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. σx, σy, σxy: global stresses. εx, εy, γxy: global strains.
	TSSIG	Target stresses/strains at Gauss points. σx, σy, σxy: global stresses. εx, εy, γxy: global strains.
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): σx, σy, σxy, σmax, σmin, β, σs, σe (see **description of principal stresses**) Strain: εx, εy, γxy, εmax, εmin, β, εs, εe
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 *Options*).	12-point (TPK6).
Mass	Default.	6-point (TPK6), 3x3 (QPK8)
	Fine (see *Options*).	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

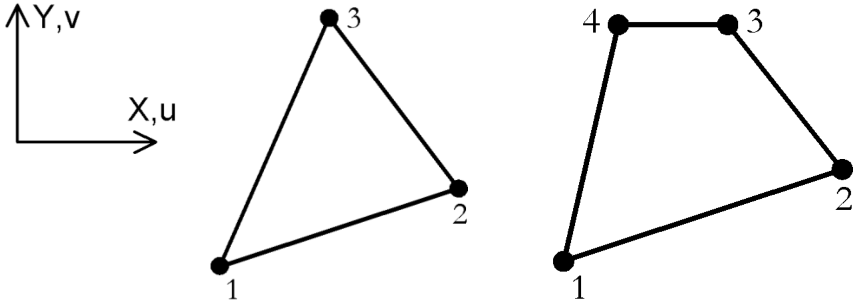
Recommendations on Use

The QPK8 and TPK6 elements are specifically designed for application to fracture mechanics problems and may 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/square root of 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	
Freedoms	U, V: at each node.
Node Coordinates	X, Y: at each node.

Geometric Properties

$t_1 \ldots t_n$	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	Not applicable.
	FLD	**Face loads**. Px, Py: local face axis pressures at nodes.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Ωx, Ωy, Ωz, αz
	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. σx, σy, σxy: global stresses. εx, εy, γxy: global strains.
	SSIG	Initial stresses/strains at Gauss points σx, σy, σxy: global stresses. εx, εy, γxy: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σx, σy, σxy: global stresses.
	SSRG	Residual stresses at Gauss points. σx, σy, σxy: global stresses.
Target Stress/Strains	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): σx, σy, σxy, σmax, σmin, β, σs, σe (see **description of principal stresses**)

Strain: εx, εy, γxy, εmax, εmin, β, εs, εe

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 *Notes*).
	Fine.	As default.
Mass	Default.	1-point (see *Notes*).
	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

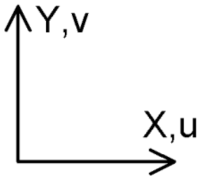
Recommendations on Use

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

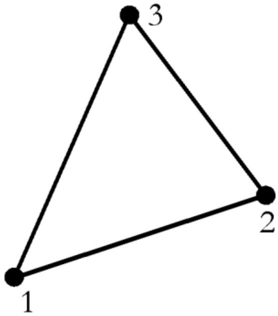
2D Plane Strain Continuum Elements

General

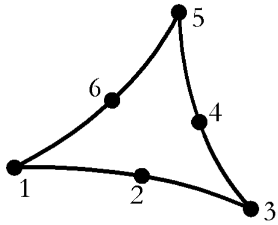
Element Name



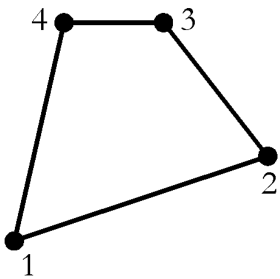
TPN3



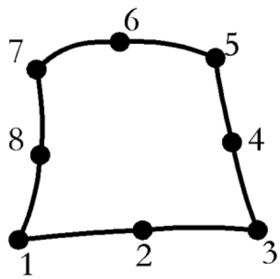
TPN6



QPN4



QPN8



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.
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.
	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 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, Ωz, αz
	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. σx, σy, σxy, σz: global stresses. εx, εy, γxy: global strains.
	SSIG	Initial stresses/strains at Gauss points. σx, σy,

		σxy, σz: global stresses. εx, εy, γxy: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σx, σy, σxy, σz: global stresses.
	SSRG	Residual stresses at Gauss points. σx, σy, σxy, σz global stresses.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. σx, σy, σxy, σz: global stresses. εx, εy, γxy: global strains.
	TSSIG	Target stresses/strains at Gauss points. σx, σy, σxy, σz: global stresses. εx, εy, γxy: global strains.
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): σx, σy, σxy, σz, σmax, σmin, β, σs, σe (see **description of principal stresses**)

Strain: εx, εy, γxy, εz =0, εmax, εmin, β, εs, εe

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 *Options*).	3x3 (QPN8), 3-point (TPN3).
Mass	Default.	1-point (TPN3), 3-point (TPN6), 2x2 (QPN4, QPN8)
	Fine (see *Options*).	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.
6. If applying an initial stress/strain or thermal load that varies across an element, a higher order element (6 or 8 nodes) should be used. A limitation of the standard isoparametric approach when used for lower order elements (3 or 4 nodes) is that only constant stress/strain fields can be imposed correctly

Restrictions

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

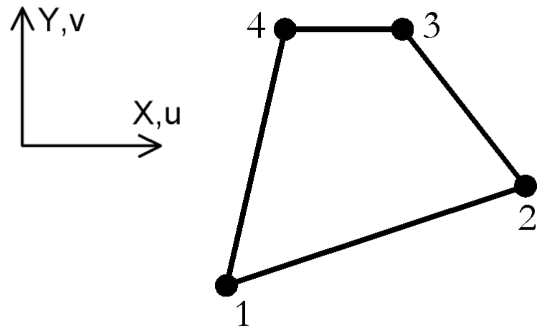
Recommendations on Use

- 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.
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.
	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)
	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	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, Ωz, αz
	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. σx, σy, σxy, σz: global stresses. εx, εy, γxy: global strains.
	SSIG	Initial stresses/strains at Gauss points. σx, σy, σxy, σz: global stresses. εx, εy, γxy: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σx, σy, σxy, σz: global stresses.

	SSRG	Residual stresses at Gauss points. σx, σy, σxy, σz global stresses.
Target Stress/Strains	TSSIE TSSIA	Target stresses/strains at nodes/for element. σx, σy, σxy, σz: global stresses. εx, εy, γxy: global strains.
	TSSIG	Target stresses/strains at Gauss points. σx, σy, σxy, σz: global stresses. εx, εy, γxy: global strains.
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): σx, σy, σxy, σz, σmax, σmin, β, σs, σe (see **description of principal stresses**)

Strain: εx, εy, γxy, εz =0, εmax, εmin, β, εs, εe

Stretch (for rubber only): V_{11}, V_{22}, V_{12}, λ_1, λ_2, λ_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 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 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

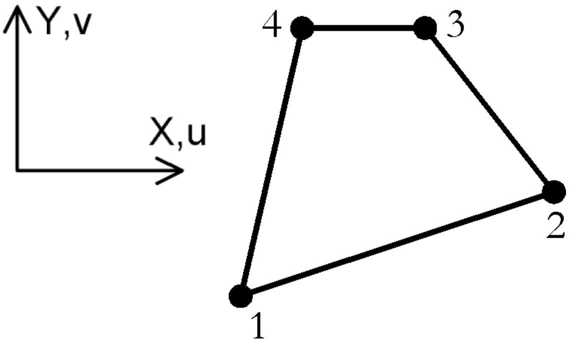
Recommendations on Use

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 Name	**QPN4L**



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
Number Of Nodes	4, 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	Not applicable	
Matrix	Not applicable	
Joint	Not applicable	
Concrete	Not applicable	
Elasto-Plastic	Implicit Optimised Von Mises	MATERIAL PROPERTIES NONLINEAR 75 (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: 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, Ωz, αz
	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. σx, σy, σxy, σz: global stresses. εx, εy, γxy: global strains.
	SSIG	Initial stresses/strains at Gauss points. σx, σy, σxy, σz: global stresses. εx, εy, γxy: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σx, σy, σxy, σz: global stresses.
	SSRG	Residual stresses at Gauss points. σx, σy, σxy,

		σz global stresses.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. σx, σy, σxy, σz: global stresses. εx, εy, γxy: global strains.
	TSSIG	Target stresses/strains at Gauss points. σx, σy, σxy, σz: global stresses. εx, εy, γxy: global strains.
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): σx, σy, σxy, σz, σmax, σmin, β, σs, σe (see **description of principal stresses**)

Principal stretches, λ_1, λ_2, λ_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).

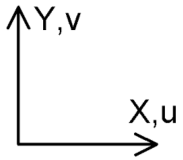
Restrictions

- Avoid excessive aspect ratio
- Avoid non-uniform initial and thermal strains with coarse meshes.

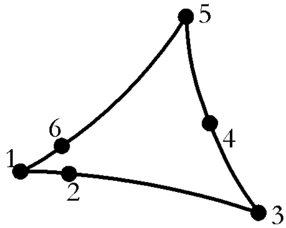
2D Plane Strain Continuum Crack Tip Elements

General

Element Name — **TNK6** / **QNK8**

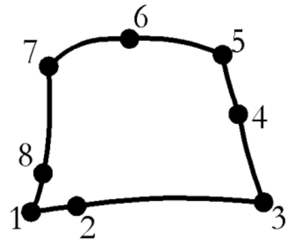


TNK6



Crack specified at Node 1

QNK8



Crack specified at Node 1

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, Ωz, αz
	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. σx,

		σy, σxy, σz: global stresses. εx, εy, γxy: global strains.
	SSIG	Initial stresses/strains at Gauss points. σx, σy, σxy, σz: global stresses. εx, εy, γxy: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σx, σy, σxy, σz: global stresses.
	SSRG	Residual stresses at Gauss points. σx, σy, σxy, σz: global stresses.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. σx, σy, σxy, σz: global stresses. εx, εy, γxy: global strains.
	TSSIG	Target stresses/strains at Gauss points. σx, σy, σxy, σz: global stresses. εx, εy, γxy: global strains.
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): σx, σy, σxy, σmax, σmin, β, σs, σe (see **description of principal stresses**)

Strain: εx, εy, γxy, εmax, εmin, β, εs, εe

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

11. 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.
12. Non-conservative loading is available with these elements when using either Updated Lagrangian or Eulerian geometric nonlinear formulations together with the FLD loading facility.

13. 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.
14. 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

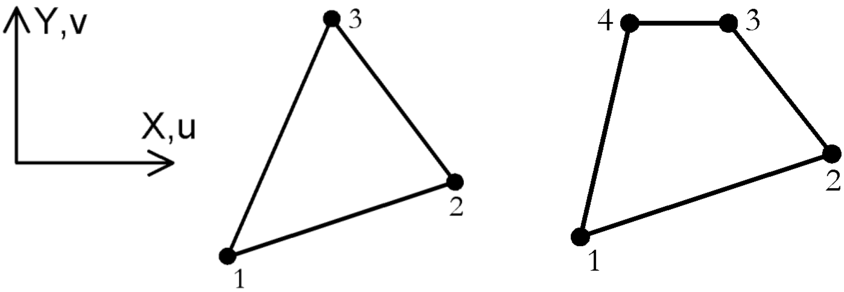
Recommendations on Use

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/square root of 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.
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:	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 *Notes*)
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. Xcbf, Ycbf, 0, 0, Ωz, αz
	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. σx, σy, σxy, σz: global stresses. εx, εy, γxy global strains.
	SSIG	Initial stresses/strains at Gauss points. σx, σy, σxy, σz: global stresses. εx, εy, γxy: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σx, σy, σxy, σz: global stresses.
	SSRG	Residual stresses at Gauss points. σx, σy, σxy, σz: global stresses.
Target Stress/Strains	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): σx, σy, σxy, σz, σmax, σmin, β, σs, σe (see **description of principal stresses**)

Strain: εx, εy, γxy, εmax, εmin, β, εs, εe

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 *Notes*).
	Fine.	As default.
Mass	Default.	1-point (see *Notes*).
	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

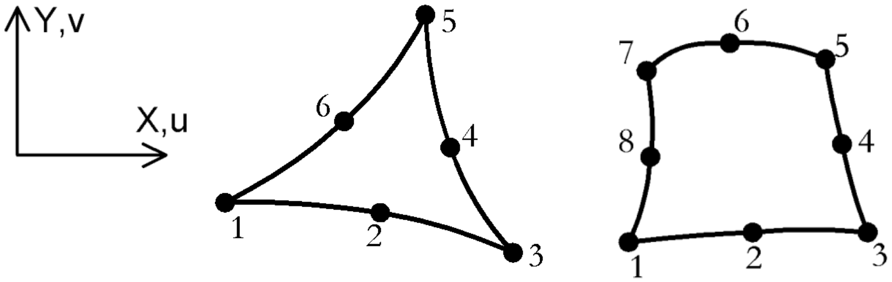
Recommendations on Use

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 Name	TPN6P	QPN8P



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.
Freedoms	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, Ωz, αz, 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. σx, σy, σxy, σz, σp global stresses. εx, εy, γxy: global strains.
	SSIG	Initial stresses/strains at Gauss points. σx, σy, σxy, σz, σp: global stresses. εx, εy, γxy: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σx, σy, σxy, σz, σp: global stresses.
	SSRG	Residual stresses at Gauss points. σx, σy, σxy, σz, σp global stresses.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. σx, σy, σxy, σz, σp global stresses. εx, εy, γxy: global strains.
	TSSIG	Target stresses/strains at Gauss points. σx, σy, σxy, σz, σp: global stresses. εx, εy, γxy: global strains.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, To, 0, 0, 0
Field Loads	Not applicable.	
Temp Dependent	Not applicable.	

Loads

LUSAS Output

Solver	Stress (default): σx, σy, σxy, σz, σp, σmax, σmin, β, σs, σe (see description of principal stresses)
	Strain: εx, εy, γxy, εz =0, εv, εmax, εmin, β, εs, εe
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

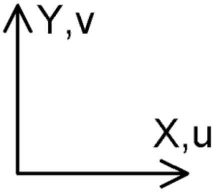
- ❑ Ensure mid-side node centrality

- ❑ Avoid excessive element curvature
- ❑ Avoid excessive aspect ratio

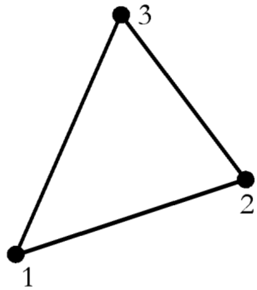
2D Axisymmetric Solid Continuum Elements

General

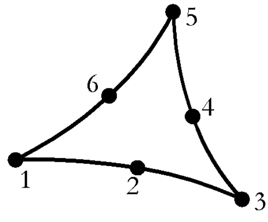
Element Name



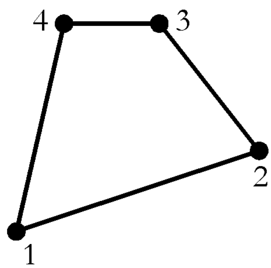
TAX3



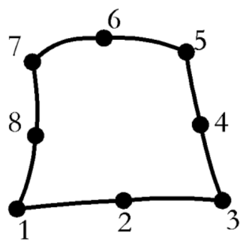
TAX6



QAX4



QAX8



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.
Freedoms	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.
	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, Ωx, Ωy (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. σx, σy, σxy, σz: global stresses. εx, εy, γxy, εz: global strains.
	SSIG	Initial stresses/strains at Gauss points. σx, σy, σxy, σz: global stresses. εx, εy, γxy, εz: global strains.

Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σx, σy, σxy, σz: global stresses.
	SSRG	Residual stresses at Gauss points. σx, σy, σxy, σz: global stresses.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. σx, σy, σxy, σz: global stresses. εx, εy, γxy, εz: global strains.
	TSSIG	Target stresses/strains at Gauss points. σx, σy, σxy, σz: global stresses. εx, εy, γxy, εz: global strains.
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): σx, σy, σxy, σz, σmax, σmin, β, σs, σe (see **description of principal stresses**)

Strain: εx, εy, γxy, εz, εmax, εmin, β, εs, εe

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 *Options*).	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 σz term as this is implicitly a principal stress in a biaxial stress field.
7. An initial stress/strain or thermal load that varies across an element should not be applied to this element. A limitation of the standard isoparametric approach when used for lower order elements is that only constant stress/strain fields can be imposed correctly.

Restrictions

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

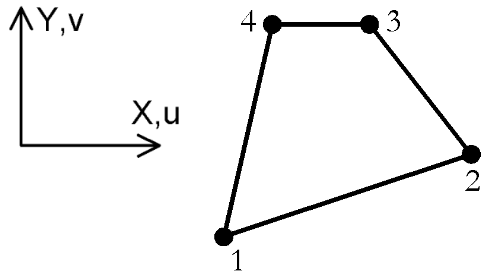
Recommendations on Use

- 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 Name	**QAX4M**



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.
Freedoms	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 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, Ωx, Ωy (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. σx, σy, σxy, σz: global stresses. εx, εy, γxy, εz: global strains.
	SSIG	Initial stresses/strains at Gauss points. σx, σy, σxy, σz: global stresses. εx, εy, γxy, εz: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σx, σy, σxy, σz: global stresses.
	SSRG	Residual stresses at Gauss points. σx, σy, σxy, σz: global stresses.

Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. σx, σy, σxy, σz: global stresses. εx, εy, γxy, εz: global strains.
	TSSIG	Target stresses/strains at Gauss points. σx, σy, σxy, σz: global stresses. εx, εy, γxy, εz: global strains.
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): σx, σy, σxy, σz, σmax, σmin, β, σs, σe (see **description of principal stresses**)

Strain: εx, εy, γxy, εz, εmax, εmin, β, εs, εe

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 σz 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.

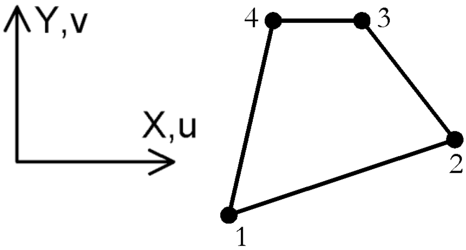
Restrictions

- Avoid excessive aspect ratio

2D Axisymmetric Solid Continuum Element for Large Strains

General

Element Name	**QAX4L**



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.
Freedoms	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 Optimised Von Mises	MATERIAL PROPERTIES NONLINEAR 75 (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, Ωx, Ωy, (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. σx, σy, σxy, σz: global stresses. εx, εy, γxy, εz: global strains.
	SSIG	Initial stresses/strains at Gauss points. σx, σy, σxy,

		σz: global stresses. εx, εy, γxy, εz: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σx, σy, σxy, σz: global stresses.
	SSRG	Residual stresses at Gauss points. σx, σy, σxy, σz: global stresses.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. σx, σy, σxy, σz: global stresses. εx, εy, γxy, εz: global strains.
	TSSIG	Target stresses/strains at Gauss points. σx, σy, σxy, σz: global stresses. εx, εy, γxy, εz: global strains.
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): σx, σy, σxy, σz, σmax, σmin, β, σs, σe (see **description of principal stresses**)

Principal stretches, λ_1, λ_2, $\lambda_3 1$, $\theta\lambda$, 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 σz term as this is implicitly a principal stress in a biaxial stress field.

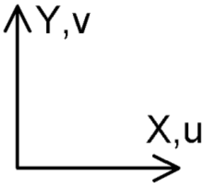
Restrictions

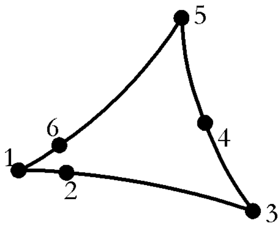
- ❑ Avoid excessive aspect ratio
- ❑ Avoid non-uniform initial and thermal strains with coarse meshes

2D Axisymmetric Solid Continuum Crack Tip Elements

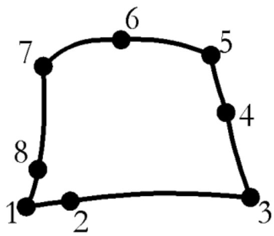
General

Element Name	**TXK6**	**QXK8**





Crack specified at Node 1



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.
Freedoms	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.
	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, Ωx, Ωy (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. σx, σy, σxy, σz: global stresses. εx, εy, γxy, εz: global strains.
	SSIG	Initial stresses/strains at Gauss points. σx, σy, σxy, σz: global stresses. εx, εy, γxy, εz: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σx, σy, σxy, σz: global stresses.
	SSRG	Residual stresses at Gauss points. σx, σy, σxy,

		σz: global stresses.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. σx, σy, σxy, σz: global stresses. εx, εy, γxy, εz: global strains.
	TSSIG	Target stresses/strains at Gauss points. σx, σy, σxy, σz: global stresses. εx, εy, γxy, εz: global strains.
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): σx, σy, σxy, σz, σmax, σmin, β, σs, σe (see **description of principal stresses**)

Strain: εx, εy, γxy, εz, εmax, εmin, β, εs, εe

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 *Options*).	12-point (TXK6).
Mass	Default.	6-point (TXK6), 3x3 (QXK8)
	Fine (see *Options*).	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

15. 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.
16. Non-conservative loading is available with these elements when using either Updated Lagrangian or Eulerian geometric nonlinear formulations together with the FLD loading facility.
17. 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.
18. Using Option 123 with local loading types, such as FLD and UDL, will cause load reversal.
19. The maximum and minimum principal stress computations for axisymmetric elements do not include the σz term as this is implicitly a principal stress in a biaxial stress field.

Restrictions

- Avoid excessive element curvature
- Avoid excessive aspect ratio

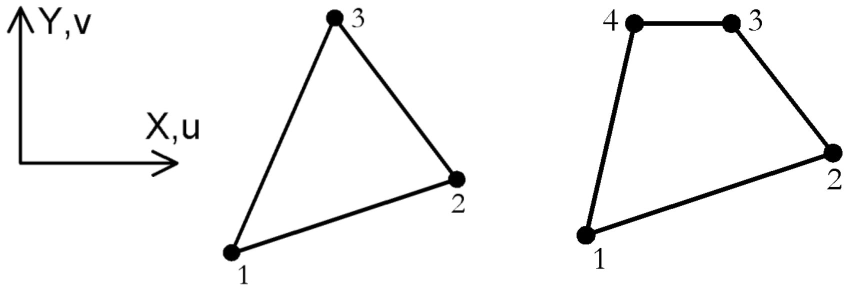
Recommendations on Use

The QXK8 and TXK6 elements are specifically designed for application to fracture mechanics problems and may 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/square root of 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.
Freedoms	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:	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 *Notes*)
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, Ωx, Ωy (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. σx, σy, σxy, σz: global stresses. εx, εy, γxy, εz: global strains.
	SSIG	Initial stress/strains at Gauss points. σx, σy, σxy, σz: global stress. εx, εy, γxy, εz: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element σx, σy, σxy, σz: global stresses.
	SSRG	Residual stresses at Gauss points. σx, σy, σxy, σz: global stresses.
Target Stress/Strains	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): σx, σy, σxy, σz, σmax, σmin, β, σs, σe (see **description of principal stresses**)

Strain: εx, εy, γxy, εz, εmax, εmin, β, εs, εe

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 *Notes*)
	Fine.	As default.
Mass	Default.	1-point (see *Notes*)
	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

20. The element formulations are based on the standard
21. The system parameter HGVISC is used to restrict element mechanisms due to under-integration. The default value is usually sufficient.
22. 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.
23. These elements must be used with a dynamic central difference scheme and a lumped mass matrix.
24. These elements are not applicable to static or eigenvalue analyses.

25. Automatic time step calculations are implemented.
26. 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.
27. If CREEP PROPERITES are defined explicit time integration must be specified in VISCOUS CONTROL.
28. Non-conservative loading is invoked when the face loading (FLD) is applied.
29. Rayleigh damping coefficients are not supported by these elements.
30. Constraint equations are not available for use with these elements.
31. 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.
32. The maximum and minimum principal stress computations for axisymmetric elements do not include the σz term as this is implicitly a principal stress in a biaxial stress field.

Restrictions

- ❑ Avoid excessive aspect ratio

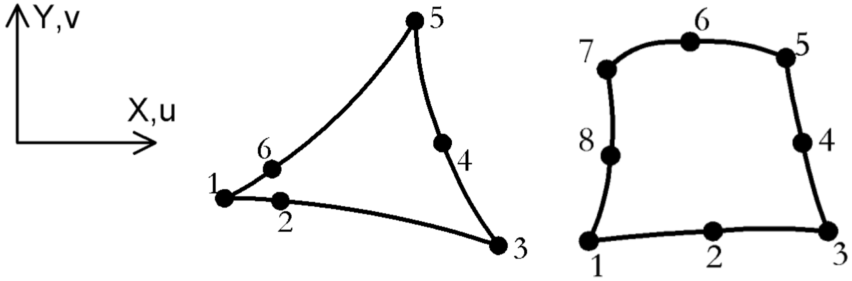
Recommendations on Use

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 Name	TAX6P	QAX8



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.
Freedoms	U, V, P: at corner nodes. U, V: at midside nodes.
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.
	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, P: at corner nodes. U, V:at midsaide 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, Ωx, Ωy (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. σx, σy, σxy, σz, σp: global stresses. εx, εy, γxy, εz: global strains.
	SSIG	Initial stresses/strains at Gauss points. σx, σy, σxy, σz, σp: global stresses. εx, εy, γxy, εz: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σx, σy, σxy, σz, σp: global stresses.

	SSRG	Residual stresses at Gauss points. σx, σy, σxy, σz, σp: global stresses.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. σx, σy, σxy, σz, σp: global stresses. εx, εy, γxy, εz: global strains.
	TSSIG	Target stresses/strains at Gauss points. σx, σy, σxy, σz, σp: global stresses. εx, εy, γxy, εz: global strains.
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): σx, σy, σxy, σz, σp, σmax, σmin, β, σs, σe (see description of principal stresses)

Strain: εx, εy, γxy, εz, εmax, εmin, β, εs, εe

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 *Options*).	3x3 (QAX8P)
Mass	Default.	3-point (TAX6P), 2x2 (QAX8P)
	Fine (see *Options*).	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 gx and gy under CBF and BFP loading.
8. The maximum and minimum principal stress computations for axisymmetric elements do not include the □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

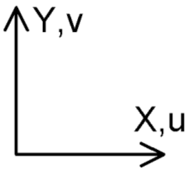
Recommendations on Use

- 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.

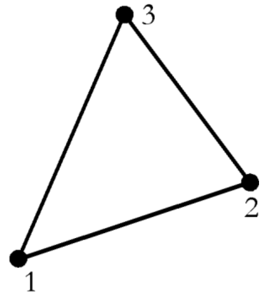
2D Axisymmetric Fourier Ring Elements

General

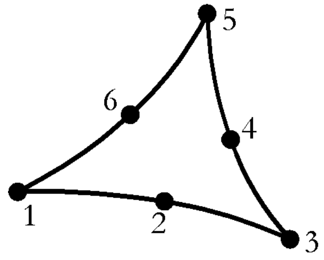
Element Name



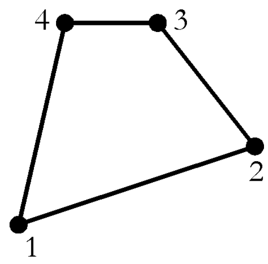
TAX3F



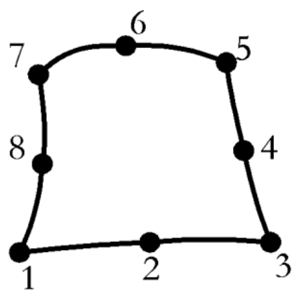
TAX6F



QAX4F



QAX8F



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.
Freedoms	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 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 (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 *Notes*). Xcbf, Ycbf, Zcbf, Ωx, Ωy Ωz, αx, αy, αz, Xo, Yo, Zo, dθ/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 Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. σx, σy, σz, σxy, σyz, σxz: local stresses. εx, εy, εz, γxy, γyz, γxz: local strains.
	SSIG	Initial stresses/strains at Gauss points. σx, σy, σz, σxy, σyz, σxz: local stresses. εx, εy, εz, γxy, γyz, γxz: local strains.
Residual Stresses	Not applicable.	
Target Stress/Strains	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): σx, σy, σz, σxy, σyz, σxz, σmax, σmin, β, σs, σe (see **description of principal stresses**)

Strain: εx, εy, εz, γxy, γyz, γxz, εmax, εmin, β, εs, εe

Use LUSAS Modeller to access results at various angles around the structure. See **Local and Global Results** in the *Modeller User Manual*

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 *Options*).	3x3 (QAX8F), 3-point (TAX3F)
Mass	Default.	1-point (TAX3F), 3-point (TAX6F), 2x2 (QAX4F, QAX8F)
	Fine (see *Options*).	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. Xo, Yo, Zo, 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 is may be omitted by setting option 102.
7. The maximum and minimum principal stress computations for axisymmetric elements do not include the σz 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

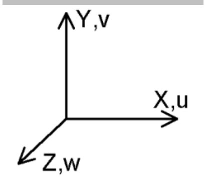
- The element is designed to model fairly solid structures, but it also performs well in comparison to standard shell analyses and may be an effective alternative for axisymmetric problems. The QAX8F is the most effective element of the family.
- If eigenvalues are required from a thin shelled structure 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.

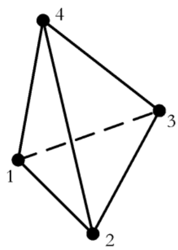
3D Solid Continuum Elements

General

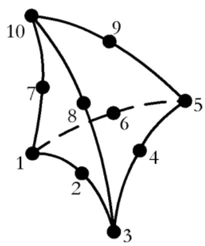
Element Name



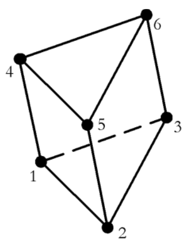
TH4



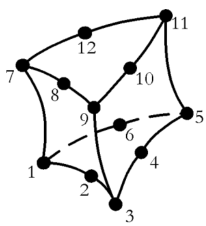
TH10



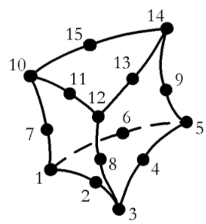
PN6



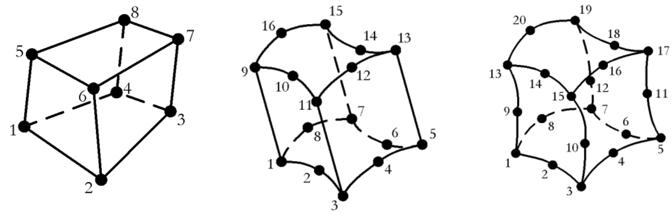
PN12



PN15



HX8 **HX16** **HX20**



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.
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, Ωx, Ωy, Ωz, αx, αy, αz
	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. σx, σy, σz, σxy, σyz, σxz: global stresses. εx, εy, εz, γxy, γyz, γxz: global strains.
	SSIG	Initial stresses/strains at Gauss points σx, σy, σz, σxy, σyz, σxz: global stresses. εx, εy, εz, γxy, γyz, γxz: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σx, σy, σz, σxy, σyz, σxz: global stresses.
	SSRG	Residual stresses at Gauss points. σx, σy, σz, σxy, σyz, σxz global stresses.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. σx, σy, σz, σxy, σyz, σxz: global stresses. εx,

		εy, εz, γxy, γyz, γxz: global strains.
	TSSIG	Target stresses/strains at Gauss points σx, σy, σz, σxy, σyz, σxz: global stresses. εx, εy, εz, γxy, γyz, γxz: global strains.
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): σx, σy, σz, σxy, σyz, σxz, σe: global stresses.

Strain: εx, εy, εz, γxy, γyz, γxz, εe: 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 *Options*).	5-point (TH10), 3x3x2 (HX16), 3x3x3 (HX20)
	Coarse (see *Options*)	13-point (HX20), 14-point (HX20)
Mass	Default.	1-point (TH4), 4-point (TH10), 3x2 (PN6, PN12, PN15), 2x2x2 (HX8, HX16, HX20)
	Fine (see *Options*).	4-point (TH4) 11-point (TH10), 14-point (TH10) 3x3x2 (HX16), 3x3x3 (HX20)
	Coarse (see *Options*)	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.

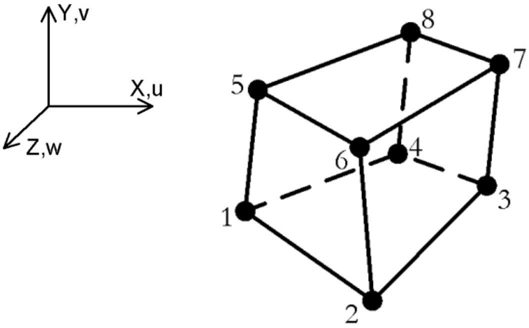
- For linear materials, the 20-noded element with a 2*2*2 Gauss rule is usually the most effective element, as this under-integration of the stiffness matrix prevents locking, i.e. over-stiff solutions will occur if the elements are used with a 3*3*3 Gauss integration rule to model structures subjected 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.
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	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	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, Ωx, Ωy, Ωz, αx, αy, αz
	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. σx, σy, σz, σxy, σyz, σxz: global stresses. εx, εy, εz, γxy, γyz, γxz: global strains.
	SSIG	Initial stresses/strains at Gauss points σx, σy, σz, σxy, σyz, σxz: global stresses. εx, εy, εz, γxy, γyz, γxz: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σx, σy, σz, σxy, σyz, σxz: global stresses.

	SSRG	Residual stresses at Gauss points. σx, σy, σz, σxy, σyz, σxz global stresses.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. σx, σy, σz, σxy, σyz, σxz: global stresses. εx, εy, εz, γxy, γyz, γxz: global strains.
	TSSIG	Target stresses/strains at Gauss points σx, σy, σz, σxy, σyz, σxz: global stresses. εx, εy, εz, γxy, γyz, γxz: global strains.
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): σx, σy, σz, σxy, σyz, σxz, σe: global stresses.

Strain: εx, εy, εz, γxy, γyz, γxz, εe: global strains.

Stretch (for rubber only): V_{11}, V_{22}, V_{33}, V_{12}, V_{23}, V_{13}, λ_1, λ_2, λ_3, det F. Where V_{ii} are components of the left stretch tensors, λ_i the principal stretches, θλ 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.

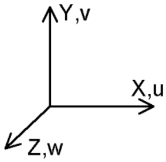
Recommendations on Use

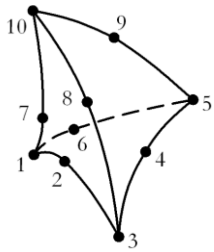
This element exhibits an improved performance when compared with the parent element HX8. The integration rules are the same as the parent element. The HX8M element does not suffer from locking due to **parasitic shear** or when the material approaches the incompressible limit. No **zero energy modes**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

3D Solid Continuum Crack Tip Elements

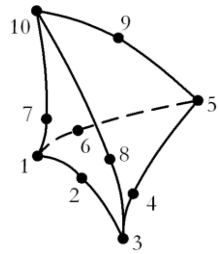
General

Element Name **TH10K**



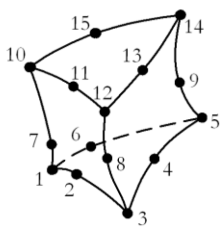


Crack specified at Node 1

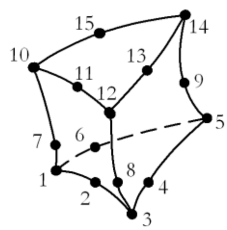


Crack specified along edge 1-2-3

PN15K

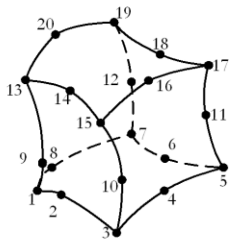


Crack specified at Node 1

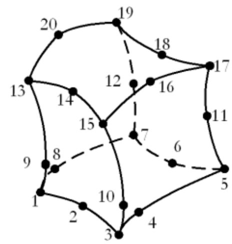


Crack specified along edge 1-2-3

HX20K



Crack specified at Node 1



Crack specified along edge 1-2-3

Element Group	3D Continuum
Element Subgroup	Solid Continuum
Element Description	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	PDSP, TPDSP	Prescribed variable. U, V, W: at each node.

Value		
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, Ωx, Ωy, Ωz, αx, αy, αz
	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. σx, σy, σz, σxy, σyz, σxz: global stresses. εx, εy, εz, γxy, γyz, γxz: global strains.
	SSIG	Initial stresses/strains at Gauss points σx, σy, σz, σxy, σyz, σxz: global stresses. εx, εy, εz, γxy, γyz, γxz: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σx, σy, σz, σxy, σyz, σxz: global stresses.
	SSRG	Residual stresses at Gauss points. σx, σy, σz, σxy, σyz, σxz global stresses.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. σx, σy, σz, σxy, σyz, σxz: global stresses. εx, εy, εz, γxy, γyz, γxz: global strains.
	TSSIG	Target stresses/strains at Gauss points σx, σy, σz, σxy, σyz, σxz: global stresses. εx, εy, εz, γxy, γyz, γxz: global strains.
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): σx, σy, σz, σxy, σyz, σxz, σe: global stresses.

Strain: εx, εy, εz, γxy, γyz, γxz, εe: 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

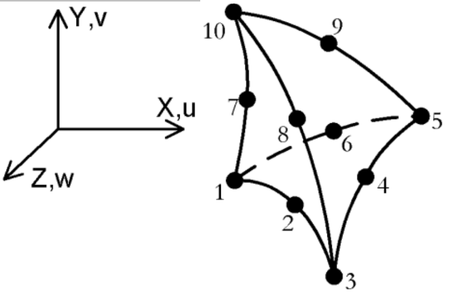
- The 3D solid crack tip elements should be used if the stress field is fully 3D, i.e. it cannot be approximated with any of the 2D crack tip elements.

- 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 over the square root of 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.
Freedoms	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		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 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, Ωx, Ωy, Ωz, αx, αy, αz
	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. σx, σy, σz, σxy, σyz, σxz: global stresses. εx, εy, εz, γxy, γyz, γxz: global strains.
	SSIG	Initial stresses/strains at Gauss points (see *Notes*). σx, σy, σz, σxy, σyz, σxz: global stresses. εx, εy, εz, γxy, γyz, γxz: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σx, σy, σz, σxy, σyz, σxz: global stresses.
	SSRG	Residual stresses at Gauss points (see *Notes*). σx, σy, σz, σxy, σyz, σxz global stresses.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. σx, σy, σz, σxy, σyz, σxz: global stresses. εx, εy, εz, γxy, γyz, γxz: global strains.
	TSSIG	Target stresses/strains at Gauss points (see *Notes*). σx, σy, σz, σxy, σyz, σxz: global stresses. εx, εy, εz, γxy, γyz, γxz: global strains.
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): σx, σy, σz, σxy, σyz, σxz: local stresses.

Strain: εx, εy, εz, γxy, γyz, γxz: 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 *Options*).	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 *Options*).	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

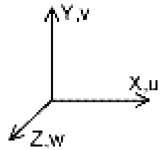
Recommendations on Use

- 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.

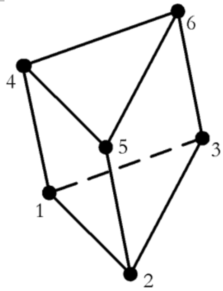
3D Solid Continuum Composite Elements (Pentahedral and Hexahedral)

General

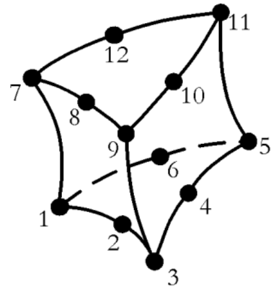
Element Name	**PN6L**	**PN12L**



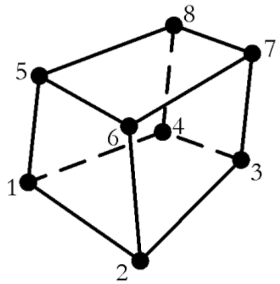
PN6L



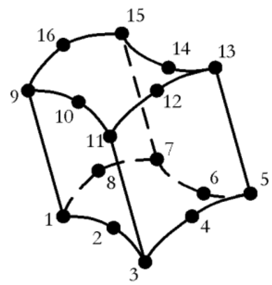
PN12L



HX8L



HX16L



Element Group	3D Continuum
Element Subgroup	Solid Continuum
Element Description	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.
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.
Freedoms	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		VISCO ELASTIC PROPERTIES
Shrinkage		SHRINKAGE CEB_FIP_90, EUROCODE_2, GENERAL, USER
Rubber	Not applicable.	
Generic Polymer		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, Ωx, Ωy, Ωz, αx, αy, αz
	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. σx, σy, σz, σxy, σyz, σxz: global stresses. εx, εy, εz, γxy, γyz, γxz: global strains.
	SSIG	Initial stresses/strains at Gauss points (see *Notes*).

		σx, σy, σz, σxy, σyz, σxz: global stresses.
		εx, εy, εz, γxy, γyz, γxz: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element.
		σx, σy, σz, σxy, σyz, σxz: global stresses.
	SSRG	Residual stresses at Gauss points (see *Notes*).
		σx, σy, σz, σxy, σyz, σxz global stresses.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element.
		σx, σy, σz, σxy, σyz, σxz: global stresses.
		εx, εy, εz, γxy, γyz, γxz: global strains.
	TSSIG	Target stresses/strains at Gauss points (see *Notes*).
		σx, σy, σz, σxy, σyz, σxz: global stresses.
		εx, εy, εz, γxy, γyz, γxz: global strains.
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): σx, σy, σz, σxy, σyz, σxz: local stresses.

Strain: εx, εy, εz, γxy, γyz, γxz: 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 *Options*).	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 *Options*).	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

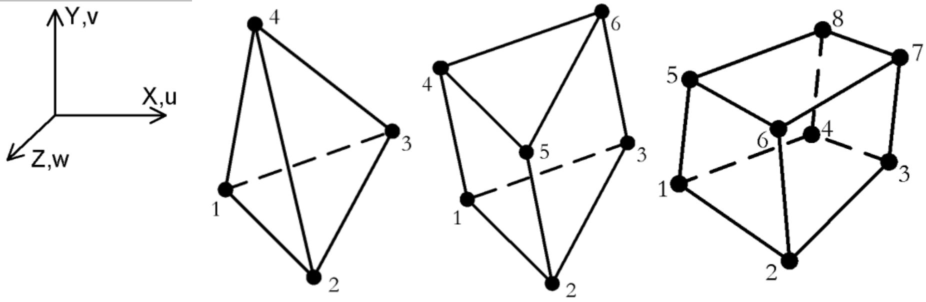
Recommendations on Use

- 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.
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:	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 *Notes*)
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	UDL	Not applicable.

Loads		
	FLD	**Face Loads**. Px, Py, Pz: local face pressures at nodes.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf, Ωx, Ωy, Ωz, αx, αy, αz
	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. σx, σy, σz, σxy, σyz, σxz: global stresses. εx, εy, εz, γxy, γyz, γxz: global strains.
	SSIG	Not applicable.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σx, σy, σz, σxy, σyz, σxz: global stresses.
	SSRG	Residual stresses at Gauss points. σx, σy, σz, σxy, σyz, σxz: global stresses.
Target Stress/Strains	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): σx, σy, σz, σxy, σyz, σxz, σe: global stresses.

Strain: not available (see *Notes*).

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 *Notes*).
	Fine.	As default.
Mass	Default.	1-point (see *Notes*).
	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

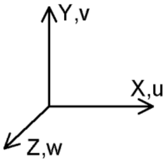
Recommendations on Use

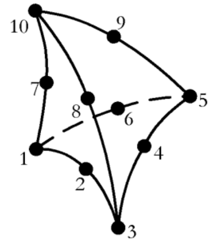
- Explicit dynamics elements may be used to define surface boundaries which will be active in a slideline analysis.
- The 3D explicit dynamics elements should be used if the stress field is fully 3D, i.e. it cannot be approximated with any of the 2D elements, e.g. a non-axisymmetric pressure vessel.

3D Solid Two Phase Continuum Elements

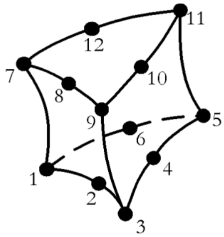
General

Element Name	TH10P

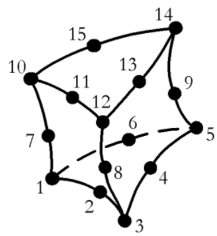




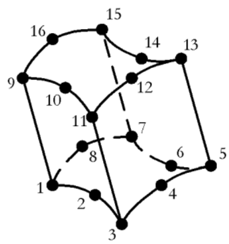
PN12P



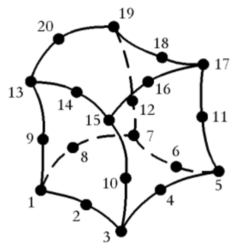
PN15P



HX16P



HX20P



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, Ωx, Ωy, Ωz, αx, αy, αz, gx, gy, gz. (See notes on use)
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, φ4, 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. σx,

		σy, σz, σxy, σyz, σxz, σp global stresses. εx, εy, εz, γxy, γyz, γxz: global strains.
	SSIG	Initial stresses/strains at Gauss points σx, σy, σz, σxy, σyz, σxz, σp: global stresses. εx, εy, εz, γxy, γyz, γxz: global strains.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element. σx, σy, σz, σxy, σyz, σxz, σp: global stresses.
	SSRG	Residual stresses at Gauss points. σx, σy, σz, σxy, σyz, σxz, σp global stresses.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. σx, σy, σz, σxy, σyz, σxz, σp global stresses. εx, εy, εz, γxy, γyz, γxz: global strains.
	TSSIG	Target stresses/strains at Gauss points σx, σy, σz, σxy, σyz, σxz, σp: global stresses. εx, εy, εz, γxy, γyz, γxz: global strains.
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): σx, σy, σz, σxy, σyz, σxz, σp, σe: global stresses.

Strain: εx, εy, εz, γxy, γyz, γxz, εv, εe: 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 *Options*).	5-point (TH10P), 3x3x2 (HX16P), 3x3x3 (HX20P)
	Coarse (see *Options*)	13-point (HX20P), 14-point (HX20P)
Mass	Default.	4-point (TH10P), 3x2 (PN12P, PN15P), 2x2x2 (HX16P, HX20P)
	Fine (see *Options*).	11-point (TH10P),14-point (TH10P), 3x3x2 (HX16P), 3x3x3 (HX20P)
	Coarse (see *Options*)	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.

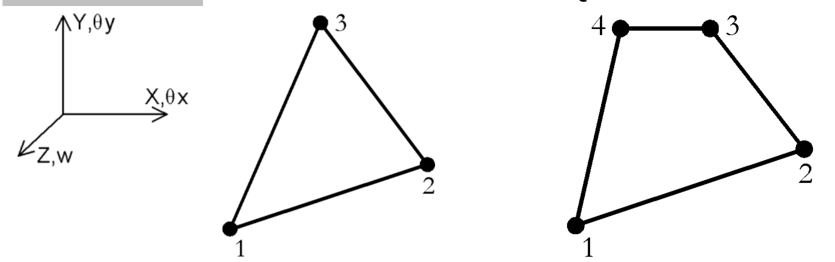
- In general, PN15P and HX20P give the best performance; TH10P is less accurate and needs to be used with a finer mesh. HX16P and PN12P should only be used to overcome connectivity problems when meshing.

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.
Freedoms	W, θx, θy: at the corner nodes.
Node Coordinates	X, Y: at each node.

Geometric Properties

t_1 ... 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 Polymer	Not applicable	
Composite	Not applicable	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. W, θx, θy: at the corner nodes.
Concentrated Loads	CL	Concentrated loads. Pz, Mx, My: at corner nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Uniformly distributed loads. Wz: normal pressure for element (global).
	FLD	Not applicable.
Body Forces	CBF	Constant body forces for element. Zcbf
	BFP, BFPE	Body force potentials at nodes/for element. φ_1, Zcbf
Velocities	VELO	Velocities. Vz: at nodes.
Accelerations	ACCE	Accelerations. Az: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. Mx, My, Mxy: moments/unit width (global). ψx, ψy, ψxy: flexural strains (global).
	SSIG	Not applicable.
Residual Stresses	Not applicable.	
Target	TSSIE, TSSIA	Target stresses/strains at nodes/for element.

Stress/Strains		Mx, My, Mxy: moments/unit width (global). ψx, ψy, ψxy: flexural strains (global).
	TSSIG	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: Mx, My, Mxy: moments/unit width (global). Strain: ψx, ψy, ψxy: 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

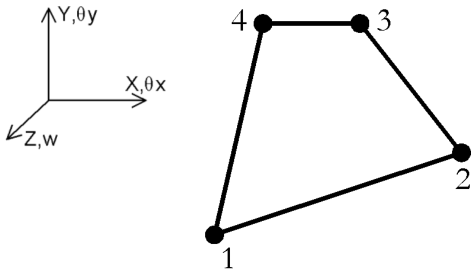
- ❑ QSL8/TSL6 elements with BSL3/BSL4/BXL4 elements.
- ❑ QTS4/TTS3 elements with BTS3 elements.

The through thickness integration is performed explicitly.

2D Isoflex Thick Plate Flexure Element

General

Element Name	**QSC4**



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.
Freedoms	W, θx, θy: at each node.
Node Coordinates	X, Y: at each node.

Geometric Properties

t_1... 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 Polymer	Not applicable
Composite	Not applicable

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. W, θx, θy: at nodes.
Concentrated Loads	CL	Concentrated loads. Pz, Mx, My: at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Uniformly distributed loads. Wz: normal pressure for element (global).
	FLD	Not applicable.
Body Forces	CBF	Constant body forces for element. Zcbf
	BFP, BFPE	Body force potentials at nodes/for element. φ_1, Zcbf
Velocities	VELO	Velocities. Vz: at nodes.
Accelerations	ACCE	Accelerations. Az: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. Mx, My, Mxy: moments/unit width (global). ψx, ψy, ψxy: flexural strains (global).
	SSIG	Not applicable.
Residual Stresses	Not applicable.	
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. Mx, My, Mxy: moments/unit width (global). ψx, ψy, ψxy: flexural strains (global).
	TSSIG	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: Mx, My, Mxy, Sx, Sy: moments, shear forces/unit width (global)

Strain: ψx, ψy, ψxy, γxz, γyz: 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

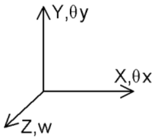
Recommendations on Use

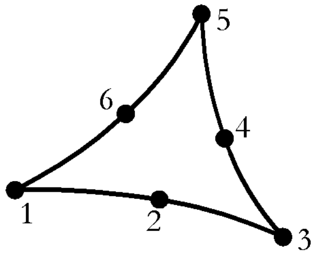
This element may be used to analyse any 2D plate type structures where transverse shear effects influence the solution, e.g. perforated thick plates.

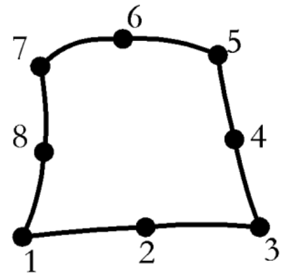
2D Mindlin Thick Plate Flexure Element

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.
Freedoms	W, θx, θy: at each node.
Node Coordinates	X, Y: at each node.

Geometric Properties

t_1... 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, θx, θy: at nodes.
Concentrated Loads	CL	Concentrated loads. Pz, Mx, My: at nodes.
Element Loads	Not applicable.	
Distributed Loads	UDL	Uniformly distributed loads. Wz: normal pressure for element (global).
	FLD	Not applicable.
Body Forces	CBF	Constant body forces for element. Zcbf
	BFP, BFPE	Body force potentials at nodes/for element. φ_1, Zcbf
Velocities	VELO	Velocities. Vz: at nodes.
Accelerations	ACCE	Accelerations. Az: at nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. Mx, My, Mxy, Sx, Sy: moments, shear forces/unit width (global). ψx, ψy, ψxy, γxz, γyz: flexural, shear strains /unit width (global).
	SSIG	Not applicable.
Residual Stresses	Not applicable.	
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. Mx, My, Mxy, Sx, Sy: moments, shear forces/unit width (global). ψx, ψy, ψxy, γxz, γyz: flexural, shear strains /unit width (global).
	TSSIG	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: Mx, My, Mxy, Sx, Sy: moments, shear forces/unit width (global). Strain: ψx, ψy, ψxy, γxz, γyz: 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 *Options*).	3x3 (QTF8).
Mass	Default.	3-point (TTF6), 2x2 (QTF8)
	Fine (see *Options*).	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

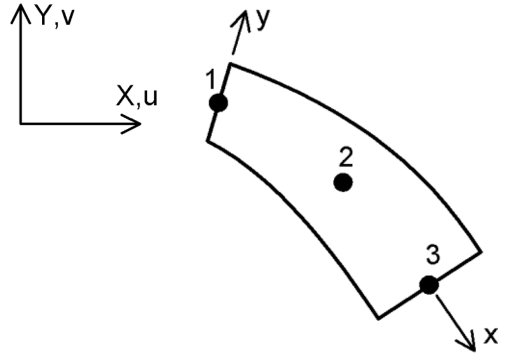
These elements may be used to analyse any 2D plate type structures where transverse shear effects influence the solution, e.g. perforated thick plates.

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	
Freedoms	U, V, θz: at end nodes. dU: (relative local in-plane displacement) at the mid-length node.
Node Coordinates	X, Y: at each node.

Geometric Properties

t1, t2, t3 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 Polymer	Not applicable	
Composite	Not applicable.	

Loading

Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V, θz: 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, Ωx, Ωy, Ωz, αz
	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	Accelerations. Ax, Ay: 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). Nx, N_θ, Mx, M_θ, 0: axial and circumferential forces, moments/unit width. εx, ε_θ, ψx, ψ_θ, 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). Nx, N_θ, Mx, M_θ,0 : axial and circumferential forces, moments/unit width. εx, ε_θ, ψx, ψ_θ, 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, (σx, σ_θ, εx, ε_θ) Bracketed terms repeated for each fibre integration point.
Residual Stresses	SSR, SSRE	Not applicable.
	SSRG	Residual stresses at Gauss points. (1) Resultants (model 29). Nx, N_θ, Mx, M_θ, 0 (2) Components (all models except 29) 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, (σx, σ_θ) Bracketed terms repeated for each fibre integration point.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. Resultants (for linear material models without cross section integration and material model 29).

		N_x, N_θ, M_x, M_θ, 0: axial and circumferential forces, moments/unit width. ε_x, ε_θ, ψ_x, ψ_θ, 0, axial and circumferential strains (all models).
	TSSIG	Target stresses/strains at Gauss points. (1) Resultants (for linear material models without cross section integration and material model 29). N_x, N_θ, M_x, M_θ,0 : axial and circumferential forces, moments/unit width. ε_x, ε_θ, ψ_x, ψ_θ, 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, (σ_x, σ_θ, ε_x, ε_θ) Bracketed terms repeated for each fibre integration point.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, dT/dy, 0, To, 0, dTo/dy, 0: in local directions.
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver Force. N_x, N_θ, M_x, M_θ: axial and circumferential forces, moments/unit width in local directions.

Strain. ε_x, ε_θ, γ_x, γ_θ: 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 *Options*).	3-point.
Mass	Default.	2-point.
	Fine (see *Options*).	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

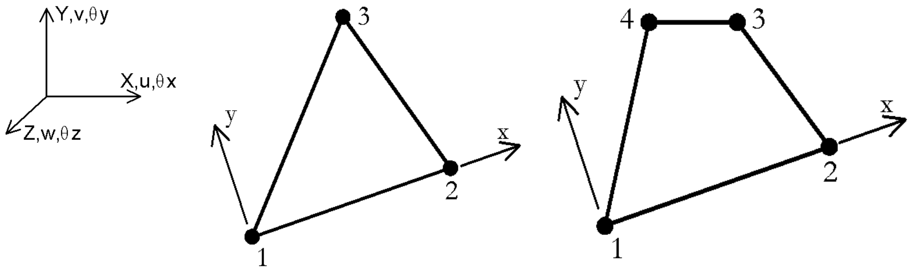
Recommendations on Use

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.
Freedoms	U, V, W, θx, θy, θz: at each node.
Node Coordinates	X, Y, Z: at each node.

Geometric Properties

Ez, t_1... t_n 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		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, θx, θy, θz: 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 Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element. Resultants. Nx, Ny, Nxy, Mx, My, Mxy: forces, moments/unit width in local directions. εx, εy, γxy, ψx, ψy, ψxy: membrane, flexural strains in local directions.
	SSIG	Not applicable.
Residual Stresses	Not applicable.	

Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. Resultants. Nx, Ny, Nxy, Mx, My, Mxy: forces, moments/unit width in local directions. εx, εy, γxy, ψx, ψy, ψxy: membrane, flexural strains in local directions.
	TSSIG	Not applicable.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, dT/dz, To, 0, 0, dTo/dz: in local directions.
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver Stress resultant: Nx, Ny, Nxy, Mx, My, Mxy: forces, moments/unit width in local directions.

Stress (default): σx, σy, σxy, σmax, σmin, β, σe: in local directions (see *Notes*).

Strain: εx, εy, γxy, ψx, ψy, ψxy: 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

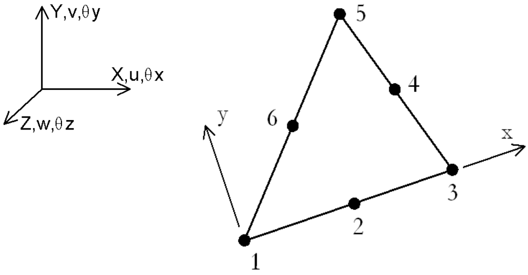
- The flat thin shell elements are suitable for modelling both flat and curved thin shell structures which exhibit negligible transverse shear deformations.
- A fine discretisation will be required to reproduce the correct behavioural response for curved structures. Therefore, the Semiloof shell elements (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 Name	**TSR6**



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.
Freedoms	U, V, W: at corner nodes. θ_1: (loof rotation) at mid-side nodes (see *Notes*).
Node Coordinates	X, Y, Z: at each node.

Geometric Properties

t_1... 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. M_1: at mid-side nodes.
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, Ωx, Ωy, Ωz, αx, αy, αz
	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. Vx, Vy, Vz: at corner nodes.
Accelerations	ACCE	Accelerations. Ax, Ay, Az: at corner nodes.
Initial Stress/Strains	SSI, SSIE	Initial stresses/strains at nodes/for element.
	SSIG	Initial stresses/strains at Gauss points. (1) Resultants (for model 29 and RIGIDITIES) Nx, Ny, Nxy, Mx, My, Mxy, εx, εy, γxy, ψx, ψy, ψxy: forces, moments/unit width and membrane/flexural strains in local directions. (2) Components (in all other cases except for nonlinear model 29 and RIGIDITIES), 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, (σx, σy, σxy, εx, εy, γxy). Bracketed terms repeat for each layer.
Residual Stresses	SSR, SSRE	Residual stresses at nodes/for element
	SSRG	Residual stresses at Gauss points. (1) Resultants (for model 29) Nx, Ny, Nxy, Mx, My, Mxy: forces, moments/unit width in local directions. (2) Components (for all nonlinear material models except model 29): 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, (σx, σy, σxy). Bracketed terms repeat for each layer.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element.
	TSSIG	Target stresses/strains at Gauss points. (1) Resultants (for model 29 and RIGIDITIES) Nx, Ny, Nxy, Mx, My, Mxy, εx, εy, γxy, ψx, ψy, ψxy: forces, moments/unit width and membrane/flexural strains in local directions. (2) Components (in all other cases except for nonlinear model 29 and RIGIDITIES), 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, (σx, σy, σxy, εx, εy, γxy). Bracketed terms repeat for each layer.
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: Nx, Ny, Nxy, Mx, My, Mxy: forces, moments/unit width in local directions.

Stress (default): σx, σy, σxy, σmax, σmin, β, σe: in local directions (see *Notes*).

Strain: εx, εy, γxy, ψx, ψy, ψxy: 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

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

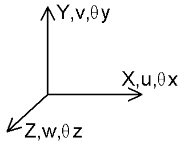
- These elements may be utilised for analysing flat and faceted 3D shell structures where the transverse shear effects do not influence the solution. The configuration of the nodal freedoms provides an element suitable for modelling intersecting shells.
- The elements are recommended for geometrically nonlinear problems where large displacements and rotations occur. The single Gauss point integration

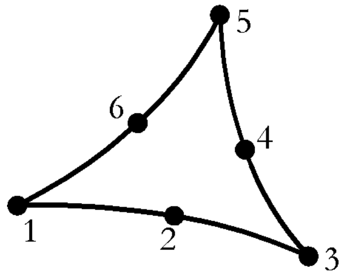
scheme gives rise to a computationally efficient solution, however, the mesh may need to be refined if there is an unacceptable differentiation in stresses between adjacent elements..

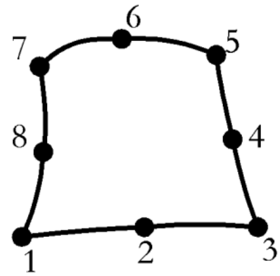
Semiloof Curved Thin Shell Elements

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 *Notes*).
Node Coordinates	X, Y, Z: at each node.

Geometric Properties

$t_1 \ldots t_n$ 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.
	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 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, M_1, M_2: at mid-side nodes.
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, Ωx, Ωy, Ωz, αx, αy, αz
	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. 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) Nx, Ny, Nxy, Mx, My, Mxy, εx, εy, γxy, ψx, ψy, ψxy: 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, (σx, σy, σxy, εx, εy, γxy) - 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) Nx, Ny, Nxy, Mx, My, Mxy: 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, (σx, σy, σxy) - with the bracketed terms repeated for each of the five layers. (See note 7 in the Notes of Use) section.
Target Stress/Strains	TSSIE, TSSIA	Not applicable.
	TSSIG	Target stresses/strains at Gauss points. (1) Resultants (for linear analysis and model 29) Nx, Ny, Nxy, Mx, My, Mxy, εx, εy, γxy, ψx, ψy, ψxy: 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, (σx, σy, σxy, εx, εy, γxy) - 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: Nx, Ny, Nxy, Mx, My, Mxy: forces, moments/unit width in local directions.

Stress (default): σx, σy, σxy, σmax, σmin, β, σe: in local directions (see *Notes*).

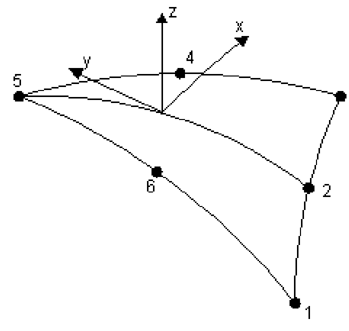
Strain: εx, εy, γxy, ψx, ψy, ψxy: membrane, flexural strains in local directions.

Modeller See **Results Tables (Appendix K)**.

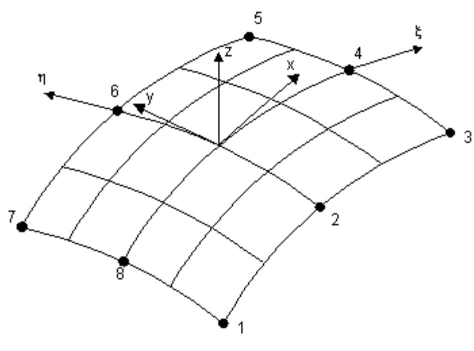
Local Axes

- **Local y axis** The local element y-axis at a point coincides with a curvilinear line ξ = 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 *Options*).	3x3 (QSL8)
	Coarse (see *Options*).	2x2 (QSL8)
Mass	Default.	3-point (TSL6), 5-point (QSL8).
	Fine (see *Options*).	3x3 (QSL8)

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

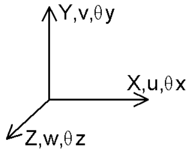
- These elements may be utilised for analysing flat and curved 3D shell structures where the transverse shear effects do not influence the solution. The configuration of the nodal freedoms provides an element suitable for modelling intersecting shells, e.g. tubular joints and also for use with solid elements (HX20).

- The elements may be combined with the Semiloof beam (BSL3,BSL4,BXL4) for analysing ribbed plates and shells.

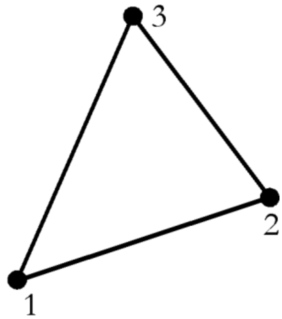
3D Thick Shell Elements

General

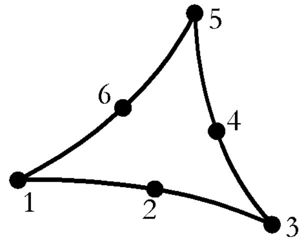
Element Name



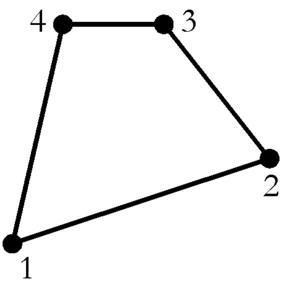
TTS3



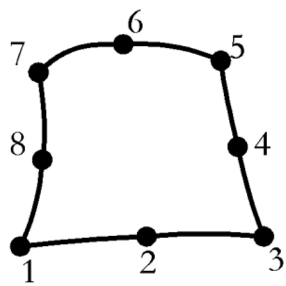
TTS6



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 *Notes*).
Number Of Nodes	3, 4, 6 or 8 numbered anticlockwise.
Freedoms	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 *Notes*). Degrees of freedom relating to global axes: U, V, W, θx, θy, θz may be enforced using the Nodal Freedom data input, or for all shell nodes by using option 278 (see *Notes*).
Node Coordinates	X, Y, Z: at each node.
Nodal Freedoms	5 or 6.

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 Polymer	Not applicable	
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, θx, θy, θz
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, Ωx, Ωy, Ωz, αx, αy, αz
	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. 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. Stress/strain components relating to local axes at Gauss points: σx, σy, σxy, σyz, σxz, εx, εy, γxy, γyz, γxz. All of these 10 terms are repeated for **each** fibre integration point through the thickness (see *Notes*).
Residual Stresses	SSR, SSRE	Not applicable.
	SSRG	Residual stresses at Gauss points. Stress components relating to local axes at Gauss points: σx, σy, σxy, σyz, σxz all of these 5 terms are repeated for **each** fibre integration point through the thickness (see *Notes*).
Target Stress/Strains	TSSIE, TSSIA	Not applicable.
	TSSIG	Target stresses/strains at Gauss points. Stress/strain components relating to local axes at Gauss points: σx, σy, σxy, σyz, σxz, εx, εy, γxy, γyz, γxz. All of these 10 terms are repeated for **each** fibre integration point through the thickness (see *Notes*).
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: Nx, Ny, Nxy, Mx, My, Mxy, Sx, Sy: forces, moments/unit width in local directions.

Stress (default): σx, σy, σxy, σyz σxz, σe: in local directions (see *Notes*).

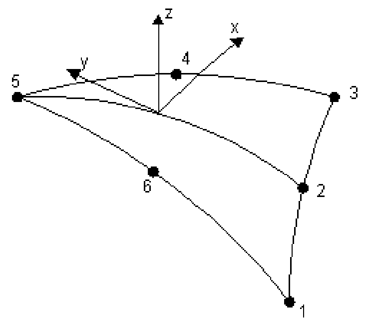
Strain: εx, εy, γxy, γyz, γxz, εe: in local directions (see *Notes*).

Modeller See **Results Tables (Appendix K)**.

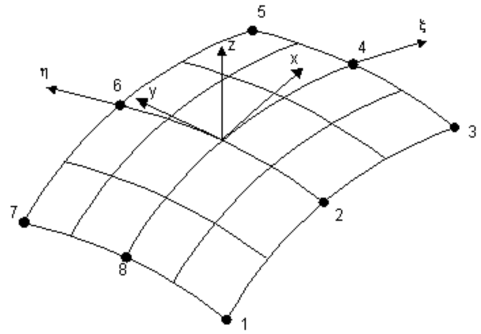
Local Axes

The local element x-axis at a point coincides with a curvilinear line η = 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 ξ = 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.
110 Use assumed shear strain field for TTS6 and QTS8 thick shell elements.
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).
417 Introduce residual bending flexibility correction for 3-node thick shell TTS3.
422 Use assumed transverse shear strain field for TTS3 thick shell element.

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.

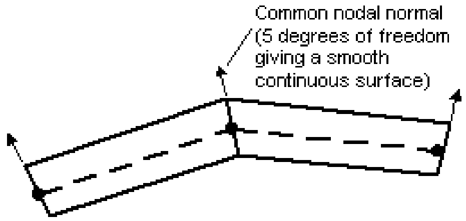
2. The QTS8 element fails the shear **patch test** when the assumed strain field is utilised with 2*2 or 5 point integration rule. When carrying out analyses

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. The introduction of assumed transverse shear strains (Option 422) significantly improves the performance of the TTS3 element. The RBF correction (Option 417) further improves the TTS3 element, especially for very thin shells. For elasto-plastic materials, the correction matrix is computed using the linear material properties.
5. 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.
6. Option 55 must be specified if nonlinear state variables are to be written to the LUSAS output file.
7. 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:

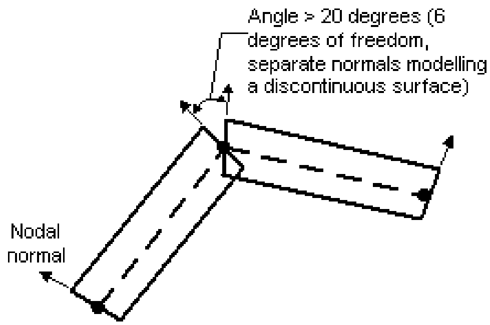


The direction of the axes defining the rotations depends upon the orientation of the normal at a node (see **Thick Shell Nodal Rotation**). In certain circumstances 3 rotational degrees of freedom relating to global axes will be assigned to a node. This is done automatically:

- When connecting with beam elements, joint elements or other types of shells, eg.QSI4.
- When a Concentrated Load is applied in LUSAS Modeller.
- When a Support is applied in LUSAS Modeller.
- When the angle between adjacent shell normals exceeds the SYSTEM parameter SHLANG (see below).
- 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 but it can be switched off in Modeller via File > Model Properties > Solution > Element options. 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. 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.

12. 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.

13. 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.

14. 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.

15. If applying an initial stress/strain or thermal load that varies across an element, a higher order element (6 or 8 nodes) should be used. A limitation of the standard isoparametric approach when used for lower order elements (3 or 4 nodes) is that only constant stress/strain fields can be imposed correctly.

Restrictions

- ❑ Ensure mid-side node centrality
- ❑ Avoid excessive element curvature

- ❑ Avoid excessive aspect ratio

Recommendations on Use

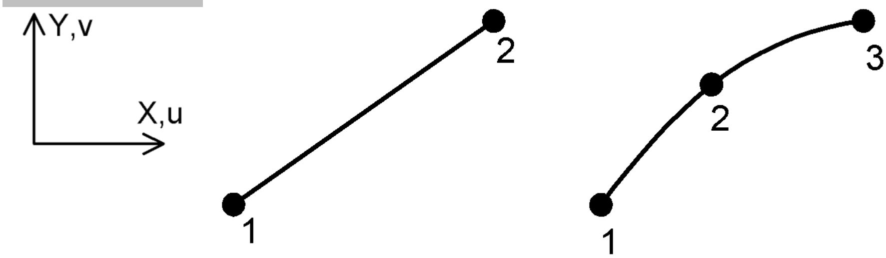
- These elements may be utilised for analysing flat and curved 3D shell structures where it is necessary to account for transverse shear. This typically involves thick shell structures where transverse shear deformation can have a considerable influence on the response. The degenerate continuum formulation 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.
Freedoms	U, V: at each node.
Node Coordinates	X, Y: at each node.

Geometric Properties

t_1... 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	Not applicable.
	FLD	**Face Loads**. Px, Py: local face pressure at nodes.
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Ωx, Ωy, Ωz, αz
	BFP, BFPE	Body force potentials at nodes/for element. 0, 0, 0, 0, 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. σx, σ_θ: axial, circumferential stress. εx, ε_θ: axial, circumferential strain.
	SSIG	Initial stresses/strains at Gauss points. σx, σ_θ: axial, circumferential stress. εx, ε_θ: axial, circumferential strain.
Residual Stresses	SSR, SSRE	Not applicable.
	SSRG	Residual stresses at Gauss points. σx, σ_θ: axial, circumferential stress.
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. σx, σ_θ: axial, circumferential stress. εx, ε_θ: axial, circumferential strain.
	TSSIG	Target stresses/strains at Gauss points. σx, σ_θ: axial, circumferential stress. εx, ε_θ: axial, circumferential strain.
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): σx, $\sigma\theta$: axial, circumferential stress.

Strain: εx, $\varepsilon\theta$: 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 *Options*).	2-point (BXM2).
Mass	Default.	1-point (BXM2), 2-point (BXM3).
	Fine (see *Options*).	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).

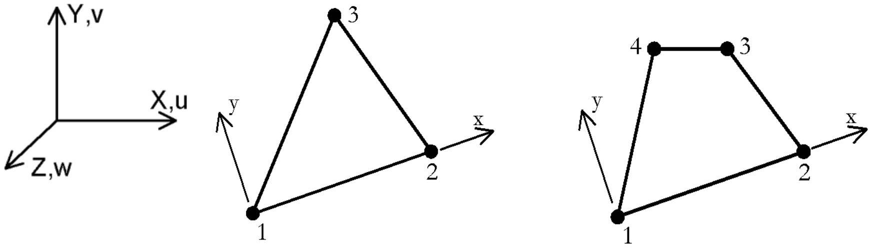
Recommendations on Use

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

Geometric Properties

t_1... 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		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, Ωx, Ωy, Ωz, αx, αy, αz
	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. εx, εy, γxy: membrane strains in local directions.
	SSIG	Initial stresses/strains at Gauss points. Nx, Ny, Nxy: forces in local directions. εx, εy, γxy: membrane strains in local directions.
Residual Stresses	Not applicable.	
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. Nx, Ny, Nxy: forces in local directions. εx, εy, γxy: membrane strains in local directions.
	TSSIG	Target stresses/strains at Gauss points. Nx, Ny, Nxy: forces in local directions. εx, εy, γxy:

		membrane strains in local directions.
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: Nx, Ny, Nxy, Nmax, Nmin, β: forces/unit length in local directions.

Stress (default): σx, σy, σxy, σmax, σmin, β: membrane stresses in local directions.

Strain: εx, εy, γxy, εmax, εmin, 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.

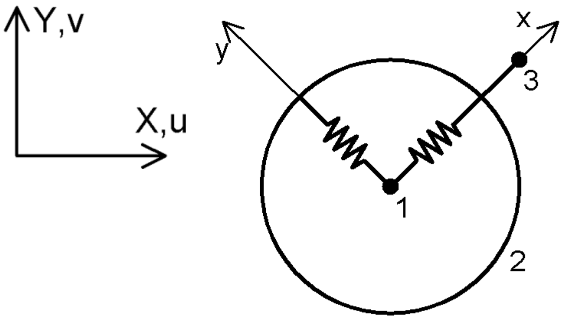
- If a structure is composed of exactly co-planar flat space membrane elements that are not stiffened by plate or shell elements, singularities may arise since there is no out-of-plane stiffness.
- If there is a possibility of bending behaviour then a thin shell should be utilised for the analysis.

Chapter 8 : Joint Elements.

2D Joint Element for Bars, Plane Stress and Plane Strain

General

Element Name	**JNT3**



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.
Freedoms	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)
	Multi-linear compound hysteresis	JOINT PROPERTIES NONLINEAR 41 2 (Joint: 2/Multi-Linear Compound Hysteresis)
	Axial force dependent multi-linear elastic	JOINT PROPERTIES NONLINEAR 42 2 (Joint: 2/Axial Force Dependent 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, Ωx, Ωy, Ωz, αz
	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: at active nodes. εx, εy: at active nodes.
	SSIG	Not applicable.
Residual Stresses	Not applicable.	
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. Fx, Fy: at active nodes. εx, εy: at active nodes.
	TSSIG	Not applicable.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T_1, T_2, T_{1o}, T_{2o}: actual and initial spring temperatures.
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Force: Fx, Fy: spring forces in local directions.
	Strain: εx, εy: 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

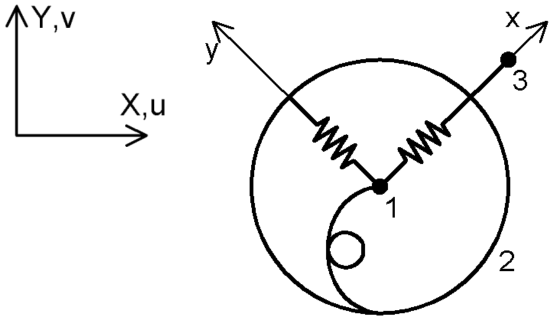
- The joint elements may be used to release degrees of freedom between elements, e.g. a hinged shell, or to provide nonlinear support conditions, e.g. friction-gap condition. Also, point masses may be represented by including a joint at an element node.

- 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.
Freedoms	U, V, θz: 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)
	Multi-linear hysteresis	JOINT PROPERTIES NONLINEAR 41 3 (Joint: 3/Multi-Linear Hysteresis)
	Multi-linear compound hysteresis	JOINT PROPERTIES

		NONLINEAR 42 3 (Joint: 3/Multi-Linear Compound Hysteresis)
	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, θz: 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, Ωx, Ωy, Ωz, αz
	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. εx, εy, ψz: strains at nodes.
	SSIG	Not applicable.
Residual Stresses	Not applicable	
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. Resultants. Fx, Fy, Mz: spring forces and moment in local directions. εx, εy, ψz: strains at nodes.
	TSSIG	Not applicable.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T_1, T_2, T_3, T_{1o}, T_{2o}, T_{3o}: actual and initial spring temperatures.
Field Loads	Not applicable	
Temp Dependent Loads	Not applicable	

LUSAS Output

Solver	Force: Fx, Fy, Mz: spring forces and moment in local directions.
	Strain: εx, εy, ψz: 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.

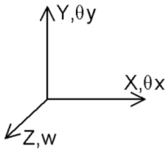
Recommendations on Use

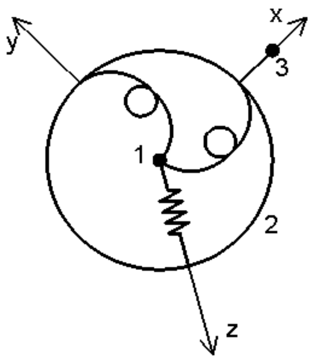
- The joint elements may be used to release degrees of freedom between elements, e.g. a hinged shell, or to provide nonlinear support conditions, e.g. friction-gap condition. Also, point masses may be represented by including a joint at an element node.
- See **Joint Element Compatibility (Appendix L)**

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.
Freedoms	W, θx, θy: 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 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:	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)
	Multi-linear compound hysteresis	JOINT PROPERTIES NONLINEAR 41 3 (Joint: 3/Multi-Linear Compound Hysteresis)
	Axial force dependent multi-linear elastic	JOINT PROPERTIES NONLINEAR 42 3 (Joint: 3/Axial Force Dependent 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, θx, θy: at active nodes.

Concentrated Loads	CL	Concentrated loads. Pz, Mx, My: 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. εz, ψx, ψy: at active nodes.
	SSIG	Not applicable.
Residual Stresses	Not applicable	
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. Fz, Mx, My: at active nodes. εz, ψx, ψy: at active nodes.
	TSSIG	Not applicable.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T_1, T_2, T_3, T_{1o}, T_{2o}, T_{3o}: actual and initial spring temperatures.
Field Loads	Not applicable	
Temp Dependent Loads	Not applicable	

LUSAS Output

Solver	Force: Pz, Mx, My: spring forces in local directions.
	Strain: εz, ψx, ψy: 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.

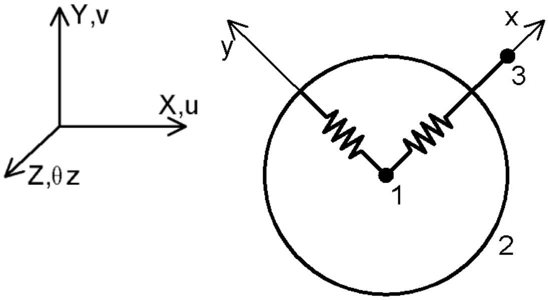
Recommendations on Use

- 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 Name	**JAX3**



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.
Freedoms	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 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)
	Multi-linear compound hysteresis	JOINT PROPERTIES NONLINEAR 41 2 (Joint: 2/Multi-Linear Compound Hysteresis)
	Axial force dependent multi-linear elastic	JOINT PROPERTIES NONLINEAR 42 2 (Joint: 2/Axial Force Dependent 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, Ωx, Ωy, Ωz, αz
	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. εx, εy: spring strains in local directions.
	SSIG	Not applicable.
Residual Stresses	Not applicable.	
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. Fx, Fy: spring forces in local directions. εx, εy: spring strains in local directions.
	TSSIG	Not applicable.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T_1, T_2, T_{1o}, T_{2o}: 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: εx, εy, εz: 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.

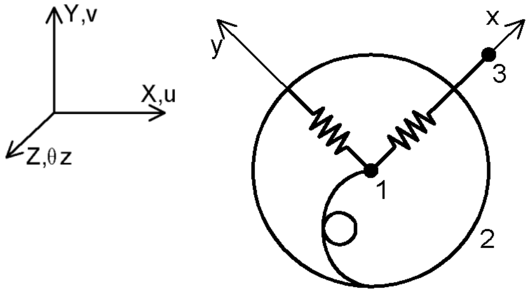
Recommendations on Use

- 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 Name	**JXS3**



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.
Freedoms	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)
	Multi-linear compound hysteresis	JOINT PROPERTIES NONLINEAR 41 3 (Joint: 3/Multi-Linear Compound Hysteresis)
	Axial force dependent multi-linear elastic	JOINT PROPERTIES NONLINEAR 42 3 (Joint: 3/Axial Force Dependent 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, 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, Ωx, Ωy, Ωz, αz
	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. εx, εy: spring strains in local directions.
	SSIG	Not applicable.
Residual Stresses	Not applicable.	
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. Fx, Fy: spring forces in local directions. εx, εy: spring strains in local directions.
	TSSIG	Not applicable.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T_1, T_2, T_3, T_{1o}, T_{2o}, T_{3o}: 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: εx, εy, εz,ψz: 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

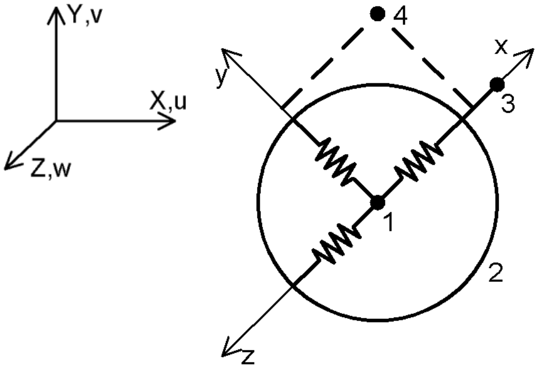
- The joint elements may be used to release degrees of freedom between elements, e.g. a hinged shell, or to provide nonlinear support conditions, e.g. friction-gap condition. Also, point masses may be represented by including a joint at an element node.

- See **Joint Element Compatibility (Appendix L)**

3D Joints for Bars, Solids and Space Membranes

General

Element Name	**JNT4**



Element Group	Joints
Element 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.
Freedoms	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)
	Multi-linear hysteresis	JOINT PROPERTIES NONLINEAR 41 3 (Joint: 3/Multi-Linear Hysteresis)
	Multi-linear compound hysteresis	JOINT PROPERTIES NONLINEAR 42 3 (Joint: 3/Multi-Linear Compound Hysteresis)
	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, Ωx, Ωy, Ωz, αx, αy, αz
	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. εx, εy, ψz: spring strains in local directions.
	SSIG	Not applicable.
Residual Stresses	Not applicable.	
Target Stress/Strains	TSSIE, TSSIA	Target initial stresses/strains at nodes/for element. Fx, Fy, Fz: spring forces in local directions. εx, εy, ψz: spring strains in local directions.
	TSSIG	Not applicable.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T_1, T_2, T_3, T_{1o}, T_{2o}, T_{3o}: 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: εx, εy, εz: 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.

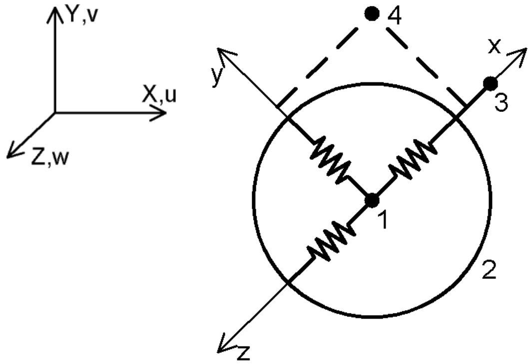
Recommendations on Use

- 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.
Freedoms	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)
	Multi-linear hysteresis	JOINT PROPERTIES NONLINEAR 41 3 (Joint: 3/Multi-Linear Hysteresis)
	Multi-linear compound hysteresis	JOINT PROPERTIES NONLINEAR 42 3 (Joint: 3/Multi-Linear Compound Hysteresis)
	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, Ωx, Ωy, Ωz, αx, αy, αz
	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. εx, εy, ψz: spring strains in local directions.
	SSIG	Not applicable.
Residual Stresses	Not applicable.	
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. Fx, Fy, Fz: spring forces in local directions. εx, εy, ψz: spring strains in local directions.
	TSSIG	Not applicable.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T_1, T_2, T_3, T_{1o}, T_{2o}, T_{3o}: 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: εx, εy, εz: 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

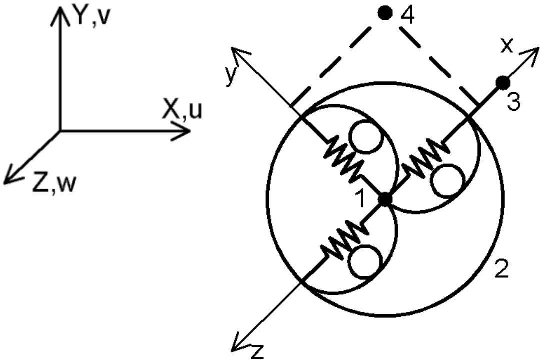
- The joint elements may be used to release degrees of freedom between elements, e.g. a hinged shell, or to provide nonlinear support conditions, e.g. friction-gap condition. Also, point masses may be represented by including a joint at an element node.

- See **Joint Element Compatibility (Appendix L)**

3D Joint Elements for Engineering, Kirchhoff and Semiloof Beams

General

Element Name	**JSH4, JL46**



Element Group	Joints
Element Subgroup	3D Joints
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.
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, θx, θy, θz: 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)
	Multi-linear hysteresis	JOINT PROPERTIES NONLINEAR 41 6 (Joint: 6/Multi-Linear Hysteresis)
	Multi-linear compound hysteresis	JOINT PROPERTIES NONLINEAR 42 6 (Joint: 6/Multi-Linear Compound Hysteresis)
	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, θx, θy, θz: 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, Ωx, Ωy, Ωz, αx, αy, αz
	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. εx, εy, εz, ψx, yy, yz: spring strains in local directions.
	SSIG	Not applicable.
Residual Stresses	Not applicable.	
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. Fx, Fy, Fz, Mx, My, Mz: spring forces in local directions. εx, εy, εz, ψx, yy, yz: spring strains in local directions.
	TSSIG	Not applicable.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T_1, T_2, T_3, T_4, T_5, T_6, T_{1o}, T_{2o}, T_{3o}, T_{4o}, T_{5o}, T_{6o}: 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: εx, εy, εz, ψx, ψy, ψz: 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

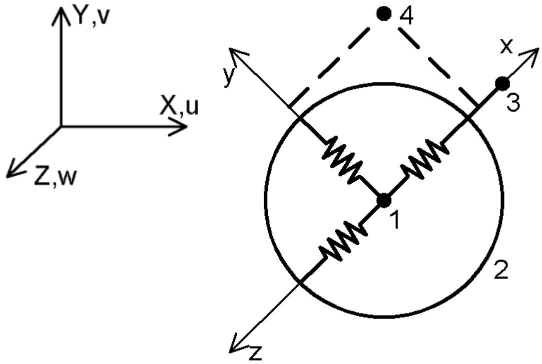
- The joint elements may be used to release degrees of freedom between elements, e.g. a hinged shell, or to provide nonlinear support conditions, e.g. friction-gap condition. Also, point masses may be represented by including a joint at an element node.

- See **Joint Element Compatibility (Appendix L)**

3D Joint Element for Semiloof Beams

General

Element Name	**JSL4**



Element Group	Joints
Element Subgroup	3D Joints
Element 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)
	Multi-linear hysteresis	JOINT PROPERTIES NONLINEAR 41 5 (Joint: 5/Multi-Linear Hysteresis)
	Multi-linear compound hysteresis	JOINT PROPERTIES NONLINEAR 42 5 (Joint: 5/Multi-Linear Compound Hysteresis)
	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, M_1, M_2: at active nodes.
Element Loads	Not applicable.	
Distributed Loads	Not applicable.	
Body Forces	CBF	Constant body forces for element. Xcbf, Ycbf, Zcbf, Ωx, Ωy, Ωz, αx, αy, αz
	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. εx, εy, εz, ψx, ψy, ψz: spring strains in local directions.
	SSIG	Not applicable.
Residual Stresses	Not applicable.	
Target Stress/Strains	TSSIE, TSSIA	Target stresses/strains at nodes/for element. Fx, Fy, Fz, Mx, My, Mz: spring forces in local directions. εx, εy, εz, ψx, ψy, ψz: spring strains in local directions.
	TSSIG	Not applicable.
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T_1, T_2, T_3, T_4, T_5, T_{1o}, T_{2o}, T_{3o}, T_{4o}, T_{5o}: actual and initial spring temperatures.
Field Loads	Not applicable.	
Temp Dependent Loads	Not applicable.	

LUSAS Output

Solver	Force: Fx, Fy, Fz, M_1, M_2: spring forces in local directions. Strain: εx, εy, εz, 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.

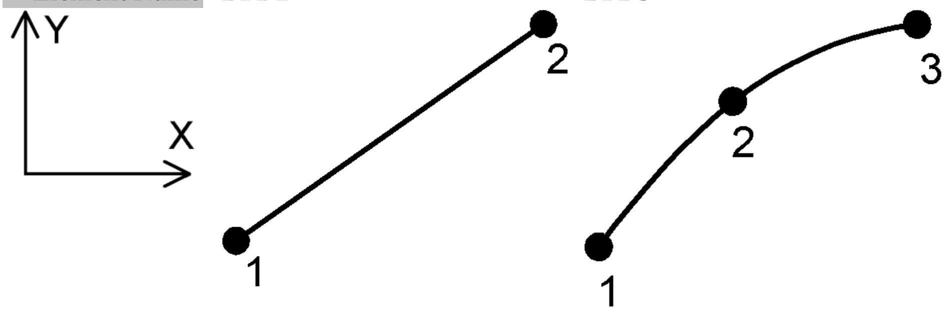
- See **Joint Element Compatibility (Appendix L)**

Chapter 9 : Thermal / Field Elements.

2D Bar Field Elements

General

Element Name	**BFD2**	**BFD3**



Element Group	Field
Element Subgroup	Thermal Bars
Element Description	Straight and curved
Number Of Nodes	2 or 3.
Freedoms	φ: field value (temperature) at each node
Node Coordinates	X, Y: at each node.

Geometric Properties

A_1 ... 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 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 (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.	
Target Stress/Strains	Not applicable.	
Temperatures	TEMP,	Temperatures at nodes/for element. T, 0, 0, 0, 0, 0, 0,

	TMPE	0 (See *Notes*.)
Field Loads	ENVT	**Environmental temperatures**. φe, hc, hr: external environmental temperature, convective and radiative heat transfer coefficients. (See *Notes*)
Temp Dependent Loads	TDET	**Temperature dependent environmental temperatures**. φe, hc, hr, T: external environmental temperature, convective and radiative heat transfer coefficients and temperature for element. (See *Notes*)
	RIHG	Internal heat generation rate. Q, T: coefficient/unit volume and temperature. (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 (BFD2), 2-point (BFD3).
	Fine (see *Options*).	2-point (BFD2), 3-point (BFD3).
Specific Heat	Default.	1-point (BFD2), 2-point (BFD3).
	Fine (see *Options*).	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

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

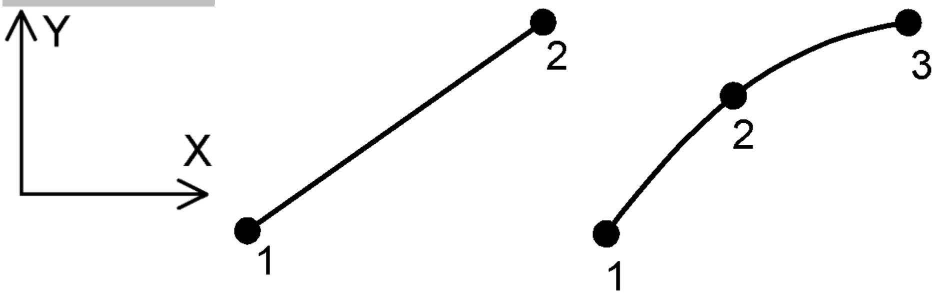
Recommendations on Use

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.
Freedoms	j: field variable (temperature) at each node.
Node Coordinates	X, Y: at each node.

Geometric Properties

t_1... 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 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 (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.	
Target Stress/Strains	Not applicable.	
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, 0, 0, 0, 0 (See *Notes*.)
Field Loads	ENVT	**Environmental temperatures**. φe, hc, hr: external environmental temperature, convective and radiative heat transfer coefficients. (See *Notes*.)
Temp Dependent Loads	TDET	**Temperature dependent environmental temperatures**. φe, hc, hr, T: external environmental temperature, convective and radiative heat transfer coefficients and temperature. (See *Notes*.)
	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 (BFX2), 2-point (BFX3).
	Fine (see *Options*).	2-point (BFX2), 3-point (BFX3).
Specific Heat	Default.	1-point (BFX2), 2-point (BFX3).
	Fine (see *Options*).	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

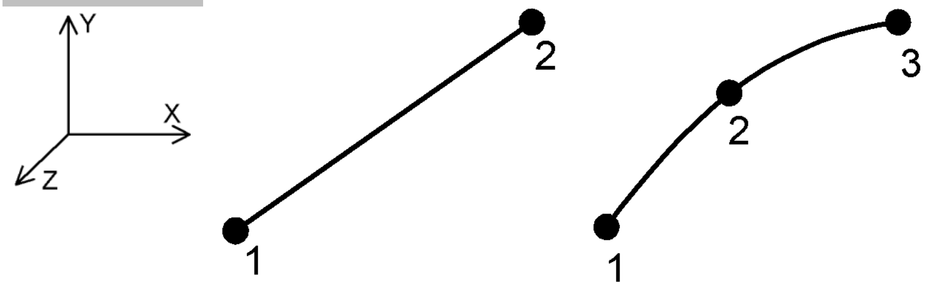
Recommendations on Use

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.
Freedoms	φ: field value (temperature) at each node
Node Coordinates	X, Y, Z: at each node.

Geometric Properties

A_1 ... 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 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 (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.	
Target Stress/Strains	Not applicable.	
Temperatures	TEMP, TMPE	Temperatures at nodes/for element. T, 0, 0, 0, 0, 0, 0, 0 (See *Notes*.)
Field Loads	ENVT	**Environmental temperatures**. φe, hc, hr: external environmental temperature, convective and radiative heat transfer coefficients. (See *Notes*.)

Temp Dependent Loads	TDET	**Temperature dependent environmental temperatures**. φe, hc, hr, T: external environmental temperature, convective and radiative heat transfer coefficients and temperature. (See *Notes*.)
	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 *Options*).	2-point (BFS2), 3-point (BFS3).
Specific Heat	Default.	1-point (BFS2), 2-point (BFS3).
	Fine (see *Options*).	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.

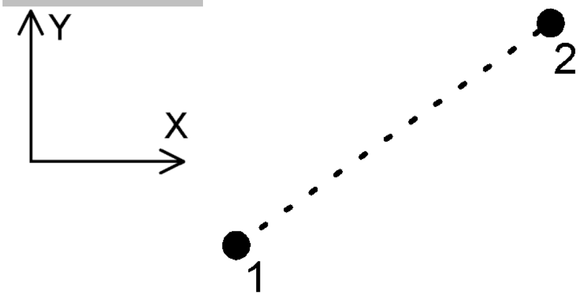
Restrictions

- ❑ Ensure mid-side node centrality
- ❑ Avoid excessive element curvature

2D Link Field Element

General

Element Name	**LFD2**



Element Group	Field
Element Subgroup	Thermal Links
Element Description	Straight conductive, convective or radiative thermal link element for 2D field analysis.
Number Of Nodes	2.
Freedoms	φ: field value (temperature) at each node.
Node Coordinates	X, Y at each node.

Geometric Properties

A_1 ... 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:	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)

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.	
Target Stress/Strains	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.

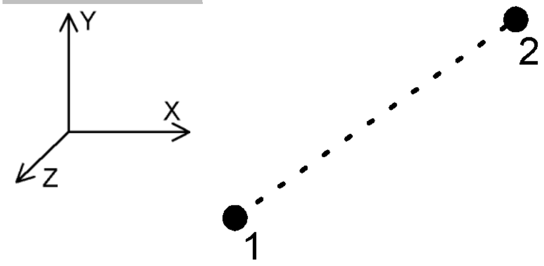
Recommendations on Use

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	
Freedoms	φ: field value (temperature) at each node.
Node Coordinates	X, Y, Z at each node.

Geometric Properties

A_1 ... 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 Potential	Not applicable.	
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.	
Target Stress/Strains	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.

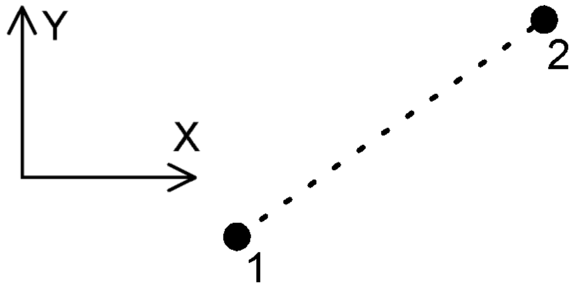
Recommendations on Use

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	
Freedoms	φ: field value (temperature) at each node.
Node Coordinates	X, Y at each node.

Geometric Properties

t_1... 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 convection/radiation:	MATERIAL PROPERTIES FIELD LINK 18 (Field: Linear Link)
Arbitrary convection/radiation:	MATERIAL PROPERTIES FIELD 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.	
Target Stress/Strains	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.

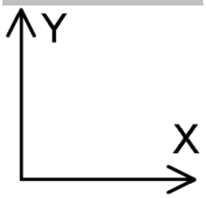
Recommendations on Use

An example of the usage of these elements is the analysis of heat conduction at contacting interfaces.

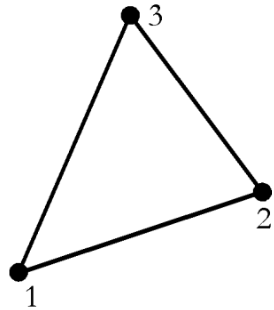
2D Plane Field Elements

General

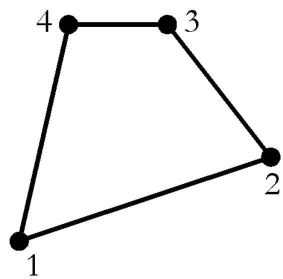
Element Name | **TFD3** | **TFD6**



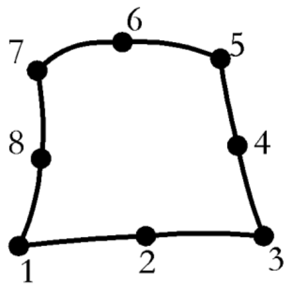
TFD3



QFD4



QFD8



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.
Freedoms	φ: field value (temperature) at each node.
Node Coordinates	X, Y: at each node.

Geometric Properties

t_1... 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 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.	
Target Stress/Strains	Not applicable.	
Temperatures	Not applicable.	
Field Loads	ENVT	**Environmental temperatures**. φe, hc, hr: external environmental temperature, convective and radiative heat transfer coefficients. (See *Notes*.)
Temp Dependent Loads	TDET	**Temperature dependent environmental temperatures**. φe, hc, hr, T: external environmental temperature, convective and radiative heat transfer coefficients and temperature. (See *Notes*.)
	RIHG	Internal heat generation rate. Q, T: coefficient/unit volume and temperature for element. (See *Notes*.)

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

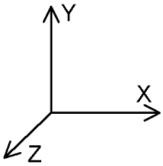
Recommendations on Use

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.

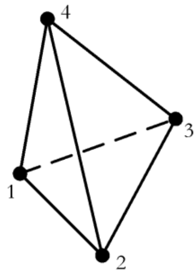
3D Solid Field Elements

General

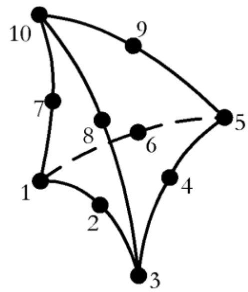
Element Name



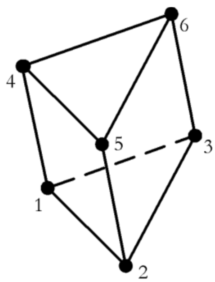
TF4



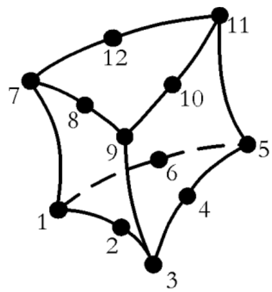
TF10



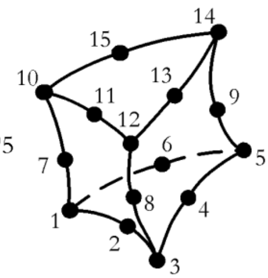
PF6



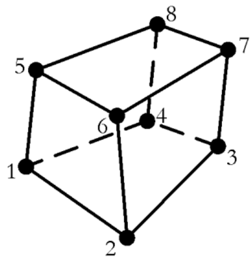
PF12



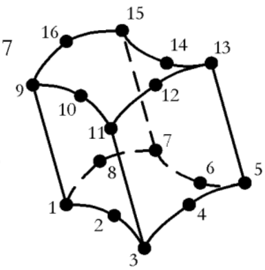
PF15



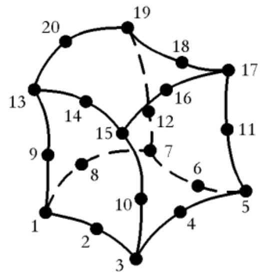
HF8



HF16



HF20



Element Group	Field
Element Subgroup	Solid Field
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.
Freedoms	φ**:** field variable 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	
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.	
Target Stress/Strains	Not applicable.	
Temperatures	Not applicable.	
Field Loads	ENVT	**Environmental temperatures**. φe, hc, hr: external environmental temperature, convective and radiative heat transfer coefficients. (See *Notes*.)
Temp Dependent Loads	TDET	**Temperature dependent environmental temperatures**. φe, hc, hr, T: external environmental temperature, convective and radiative heat transfer coefficients and temperature. (See *Notes*.)
	RIHG	Internal heat generation rate. Q, T: coefficient/unit volume and temperature for element. (See *Notes*.)

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 *Options*).	*5*-point (TF10) 3x3x2 (HF16), 3x3x3 (HF20)
	Coarse (see *Options*).	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 *Options*).	*5*-point (TF10) 3x3x2 (HF16), 3x3x3 (HF20)
	Coarse (see *Options*).	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

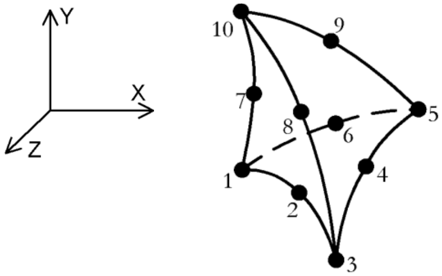
Recommendations on Use

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 Name **TF10S**



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 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.	
Target	Not	

Stress/Strains	applicable.	
Temperatures	Not applicable.	
Field Loads	ENVT	Environmental temperatures φe, hc, hr: external environmental temperature, convective and radiative heat transfer coefficients. (See *Notes*.)
Temp Dependent Loads	TDET	Temperature dependent environmental temperatures. φe, hc, hr, T: external environmental temperature, convective and radiative heat transfer coefficients and temperature. (See *Notes*.)
	RIHG	Internal heat generation rate. Q, T: coefficient/unit volume and temperature for element. (See *Notes*.)

LUSAS Output

Solver Field variable (temperature). gx, gy, gz, qx, qy, qz: 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 Notes), 3-point for a pentahedral/pyramid subdivision, 2x2 for a hexahedral/wrick subdivision
	Fine (see *Options*).	1-point for a tetrahedral subdivision (see Notes), 3x2 for a pentahedral/pyramid subdivision, 2x2 x2

		for a hexahedral/wrick subdivision
Specific Heat	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 *Options)*.	11-point or (see Options) 14 -point for the whole 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

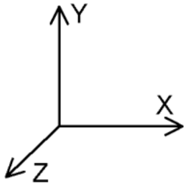
Recommendations on Use

- 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.

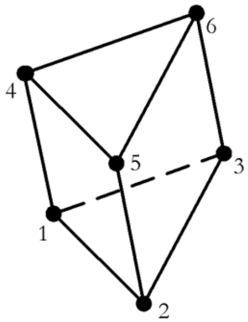
3D Solid Composite Field Elements (Pentahedral and Hexahedral)

General

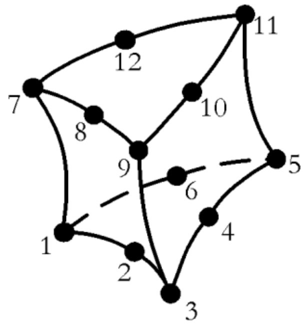
Element Name



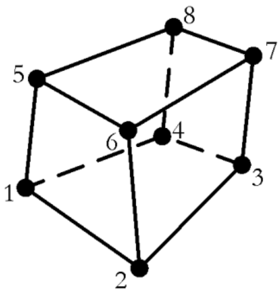
PF6C



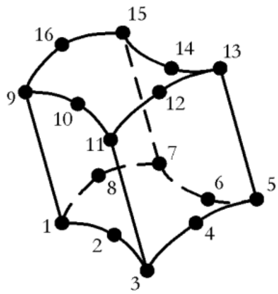
PF12C



HF8C



HF16C



Element Group	Field
Element Subgroup	Solid Field
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.
Freedoms	**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.	
Target Stress/Strains	Not applicable.	
Temperatures	Not applicable.	
Field Loads	ENVT	Environmental temperatures φe, hc, hr: external environmental temperature, convective and radiative heat transfer coefficients. (See *Notes*.)
Temp Dependent Loads	TDET	Temperature dependent environmental temperatures. φe, hc, hr, T: external environmental temperature, convective and radiative heat transfer coefficients and temperature. (See *Notes*.)
	RIHG	Internal heat generation rate. Q, T: coefficient/unit volume and temperature for element. (See *Notes*.)

LUSAS Output

Solver Field variable (temperature). gx, gy, gz, qx, qy, qz: 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 *Options*).	3-point for each layer (PF6C), 3x3 for each layer (HF16C)
Specific Heat	Default.	3x2 for the whole element (PF6C, PF12C) or (see Options) 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 *Options)*.	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

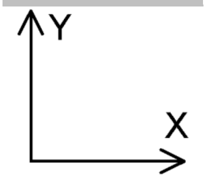
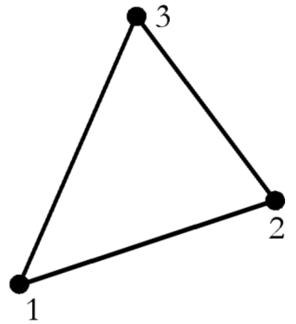
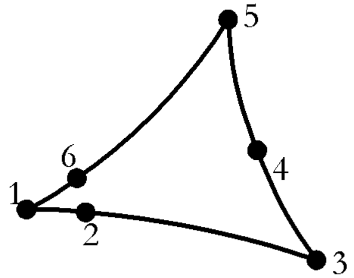
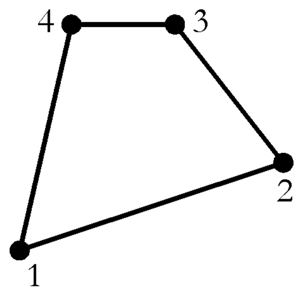
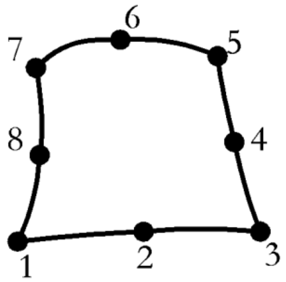
- Ensure mid-side node centrality
- Avoid excessive element curvature
- Avoid excessive aspect ratio
- Constant layer thickness for each individual layer

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 Name	TXF3	TXF6
		
	QXF4	**QXF8**
		

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.
Freedoms	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.	
Target Stress/Strains	Not applicable.	
Temperatures	Not applicable.	
Field Loads	ENVT	**Environmental temperatures**. φe, hc, hr: external environmental temperature, convective and radiative heat transfer coefficients. (See *Notes*.)
Temp Dependent Loads	TDET	**Temperature dependent environmental temperatures**. φe, hc, hr, T: external environmental temperature, convective and radiative heat transfer coefficients and temperature. (See *Notes*.)
	RIHG	Internal heat generation rate. Q, T: coefficient/unit volume and temperature for element. (See *Notes*.)

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 *Options*).	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

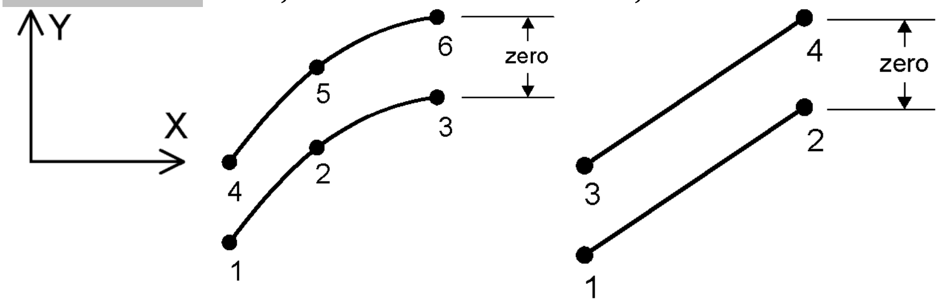
The axisymmetric field elements are suitable for analysing solid field problems which exhibit geometric and loading symmetry about a given axis, e.g. temperature distribution in a pipe or radial groundwater flow into a well.

Chapter 10 : Interface Elements.

2D Interface Element

General

Element Name IPN6, IAX6 IPN4, IAX4



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	MATERIAL PROPERTIES NONLINEAR 25
Rubber	Not applicable	
Generic Polymer	Not applicable	
Composite	Not applicable	

Loading

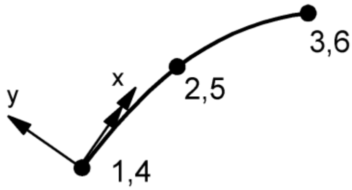
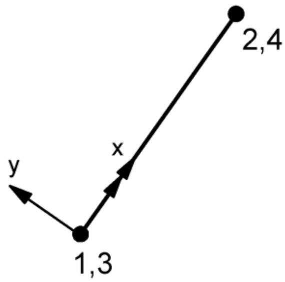
Prescribed Value	PDSP, TPDSP	Prescribed variable. U, V: at each node.
Concentrated Loads	CL	Concentrated loads. Px, Py: at each node.
Element Loads	Not applicable.	
Distributed Loads	Not applicable.	
Body Forces	Not applicable.	
Velocities	VELO	Velocities. Vx, Vy: at nodes.
Accelerations	ACCE	Acceleration Ax, Ay: at nodes.
Initial Stress/Strains	Not applicable.	
Residual Stresses	Not applicable.	
Target Stress/Strains	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	IPN4,IAX4
Evaluated at each node.		

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 (DZ_3 - DZ_6) > 0, where DZ_i 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.

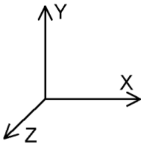
Recommendations on Use

These elements should be used at places of potential delamination between 2D plane and axisymmetric continuum elements.

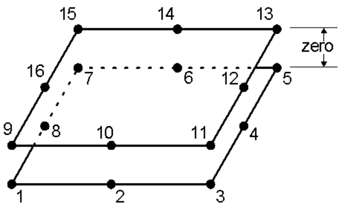
3D Interface Element

General

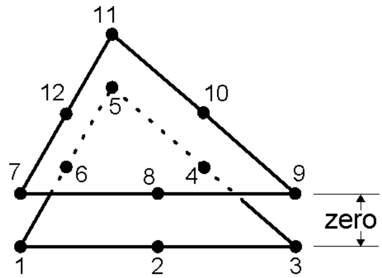
Element Name



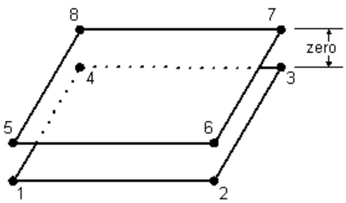
IS16



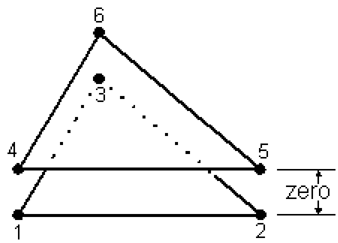
IS12



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
Freedoms	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	

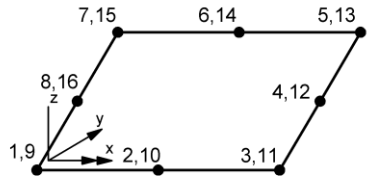
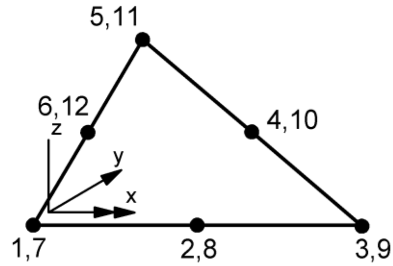
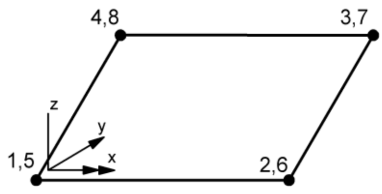
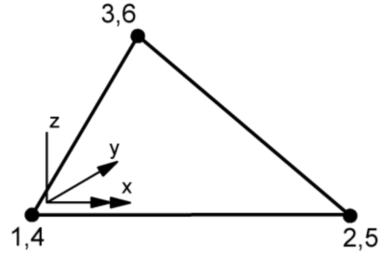
Loading

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.	
Target Stress/Strains	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	IS16	IS12
Evaluated at each node.		
	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

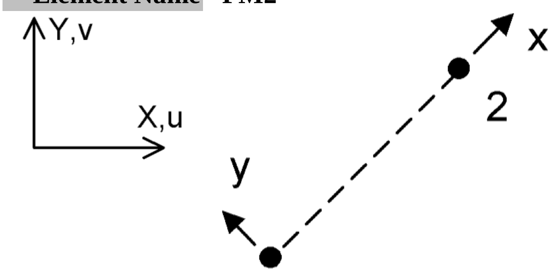
These elements should be used at places of potential delamination between 3D continuum elements.

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 2nd node is used to define the local x-axis.
Freedoms	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.

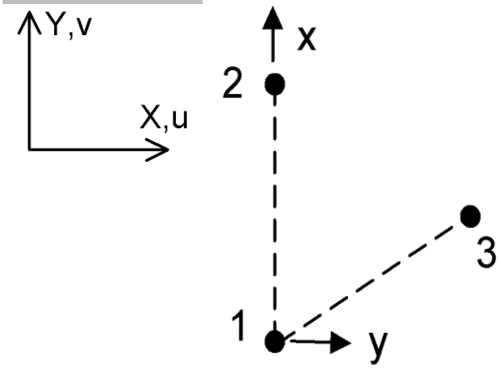
Recommendations on Use

The 2D point mass element can be used to model point masses occur in a 2D structure.

3D Point Mass Element

General

Element Name	**PM3**



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.
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 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.

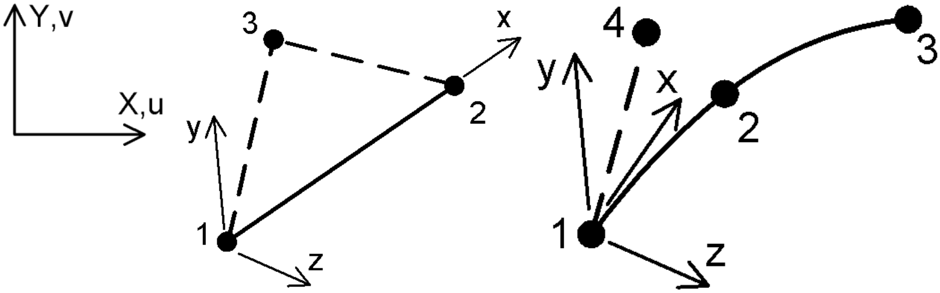
Recommendations on Use

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 3rd node is used to define the local x-y plane. 4 (LMS4). The 4th 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

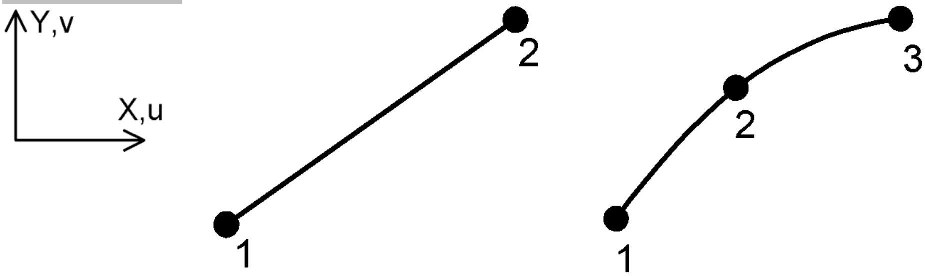
Recommendations on Use

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	
Freedoms	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

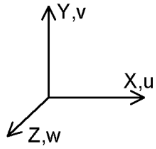
Recommendations on Use

2D line mass elements can be used to model masses along an edge in a 2D structure.

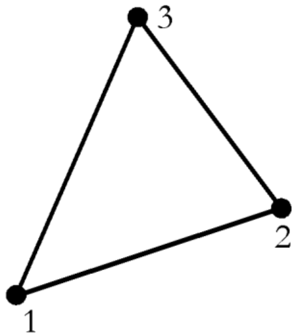
Surface Mass Elements

General

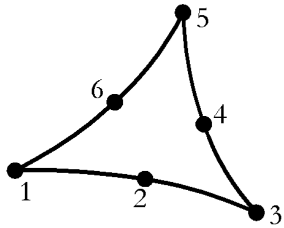
Element Name



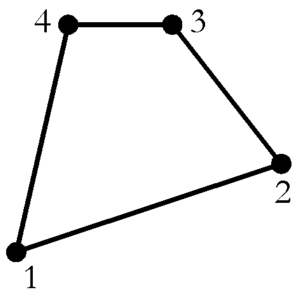
TM3



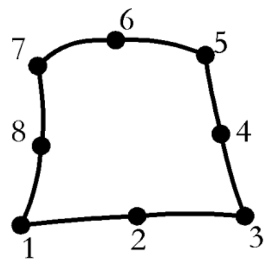
TM6



QM4



QM8



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

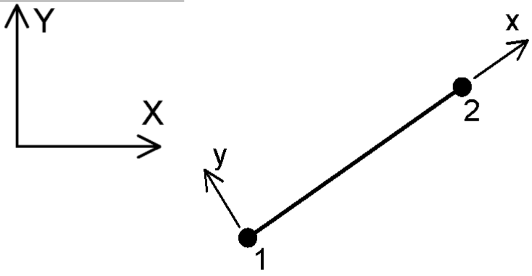
The surface mass elements can be used to model masses on a surface 3D structures.

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
Freedoms	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.

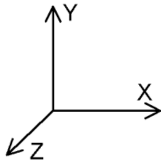
Recommendations on Use

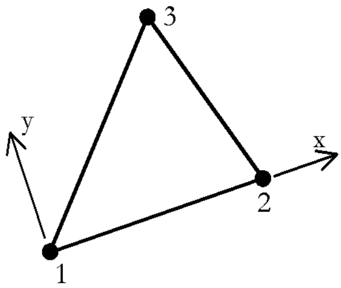
These elements should be used when one of the surfaces which come into contact is non-deformable. Using these elements will make the analysis faster.

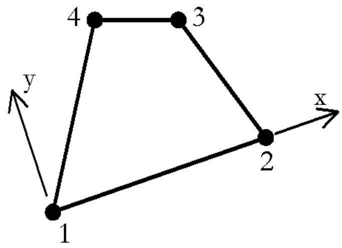
Rigid Surface 3D Elements

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
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)

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.

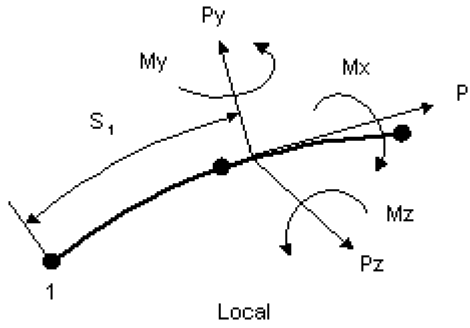
ELDS Element Loads

These are referred to as Internal Beam Point Loads and Internal Beam Distributed Loads within LUSAS Modeller.

Parameter	Description
Itype	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

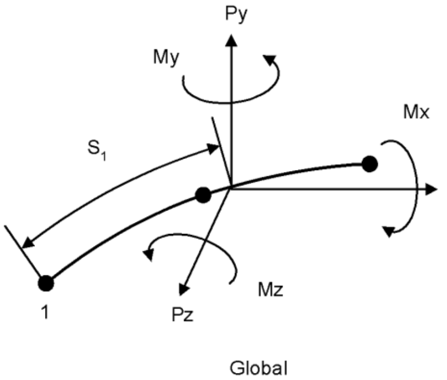
Itype 11

Point loads and moments in local directions



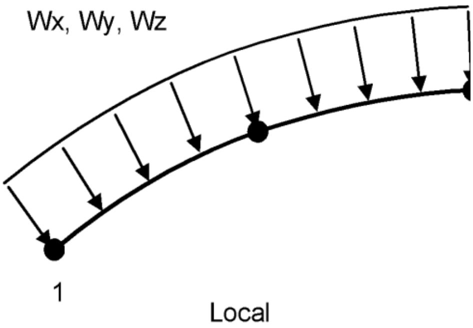
Itype 12

Point loads and moments in global directions



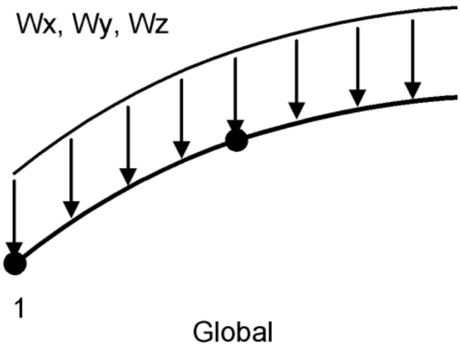
Itype 21

Uniformly distributed loads in local directions



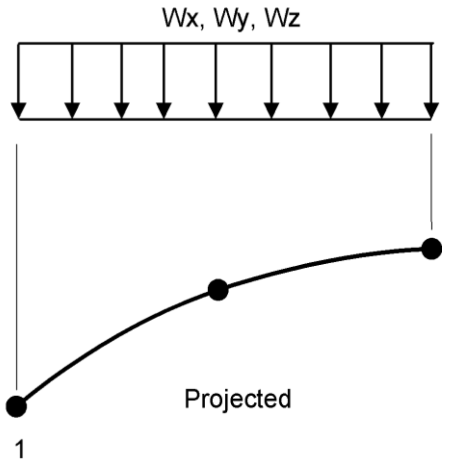
Itype 22

Uniformly distributed loads in global directions



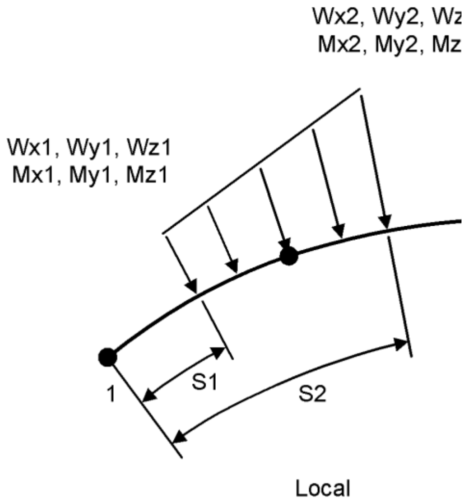
Itype 23

Uniformly distributed projected loads in global directions



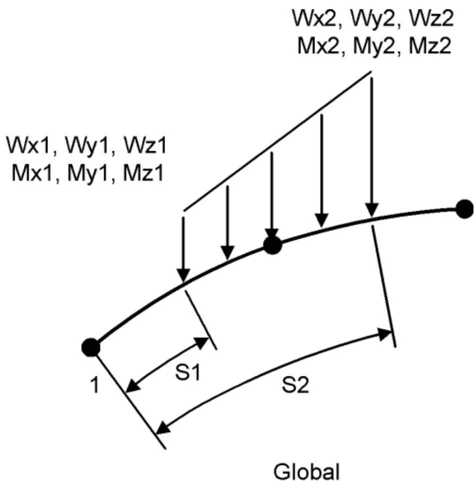
Itype 31

Distributed loads in local directions. Multiple load sets supported.



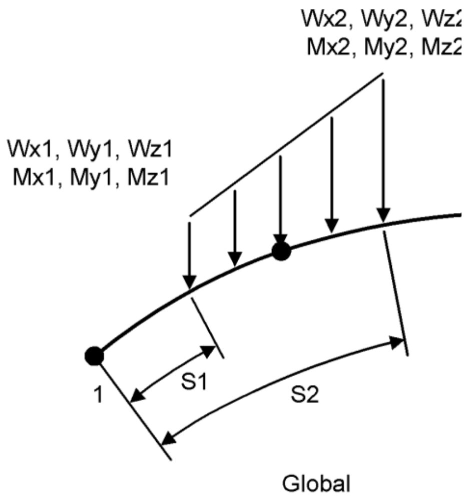
Itype 32

Distributed loads in global directions. Multiple load sets supported.



Itype 33

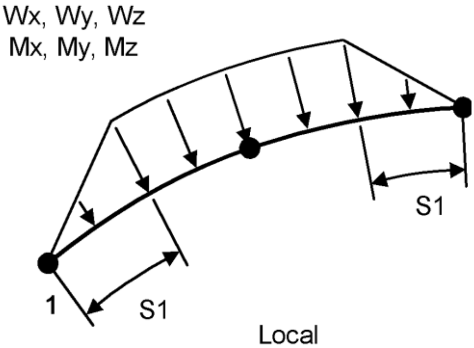
Distributed projected loads in global directions. Multiple load sets supported.



ltype 41

Trapezoidal loads in local directions

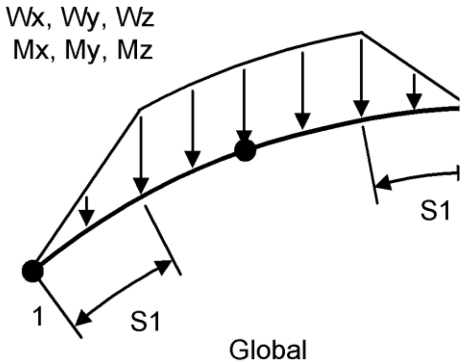
Definition only supported in LUSAS Solver. In LUSAS Modeller trapezoidal beam loads are defined in accordance with ltype 31.



ltype 42

Trapezoidal loads in global directions

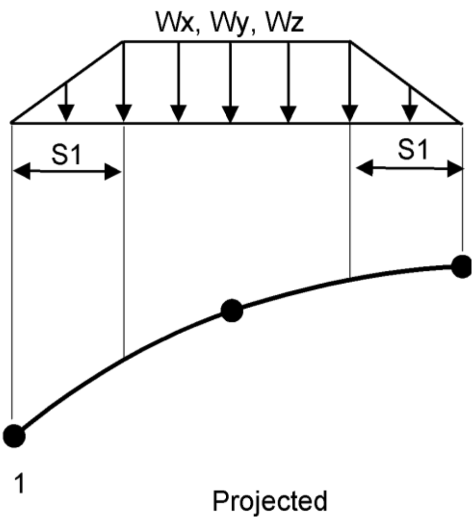
Definition only supported in LUSAS Solver. In LUSAS Modeller trapezoidal beam loads are defined in accordance with ltype 32.



ltype 43

Trapezoidal projected loads in global directions

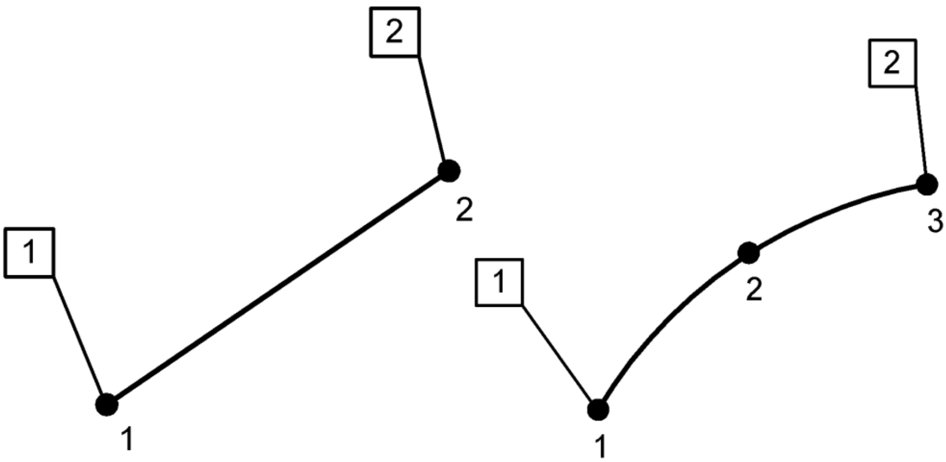
Definition only supported in LUSAS Solver. In LUSAS Modeller trapezoidal beam loads are defined in accordance with ltype 33.



ENVT/TDET Environmental Temperature Loading

Contains some or all of:

Parameter	Description
je	External environmental temperature.
hc	Convective heat transfer coefficient.
hr	Radiative heat transfer coefficient.
T	Temperature for element.

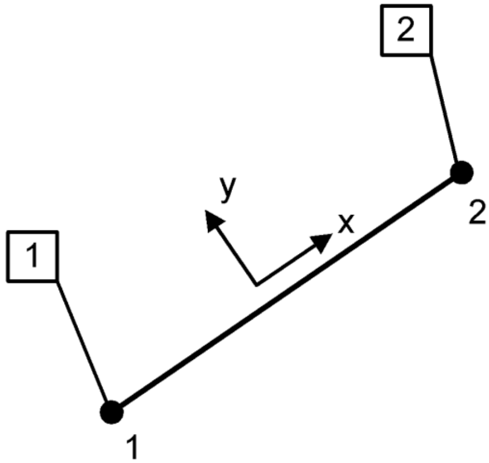


Face Numbering Convention for Thermal Bars

Note

The environmental temperature loading for node 2 cannot be specified for a 3 noded bar.

FLD Face loading applied to thermal bars



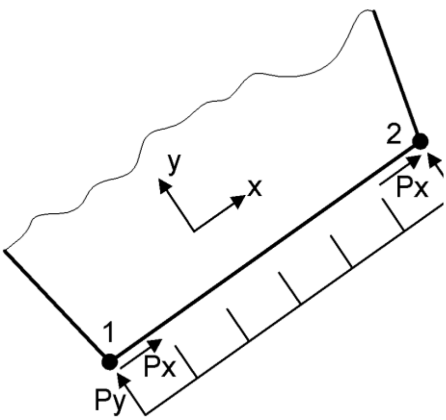
Face number = local node number

Face Numbering Convention for Thermal Bars

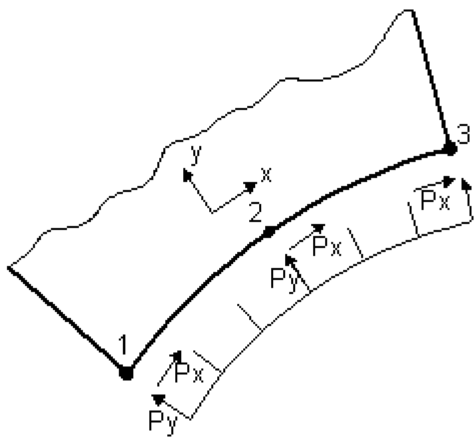
Face Loads On 2D Continuum Elements

Parameter	Description
Px, Py	Face pressures defined at nodes in local x, y directions

2-Noded Element Faces



3-Noded Element Faces

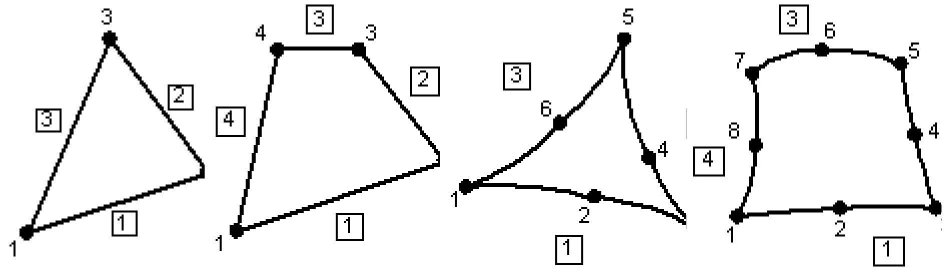


Note

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**



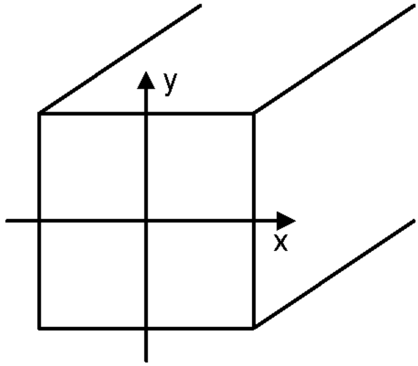
Face Loads On 3D Continuum Elements

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



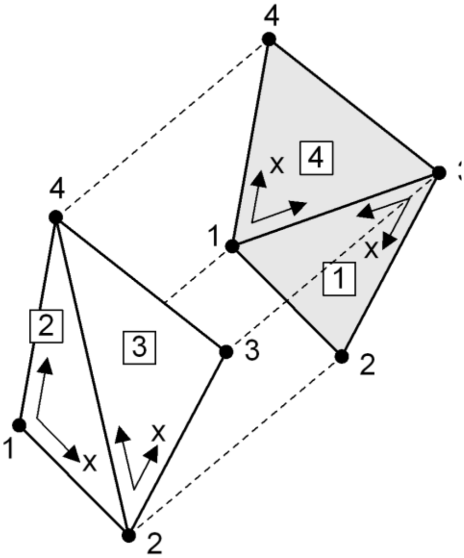
Face Numbering Convention

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.

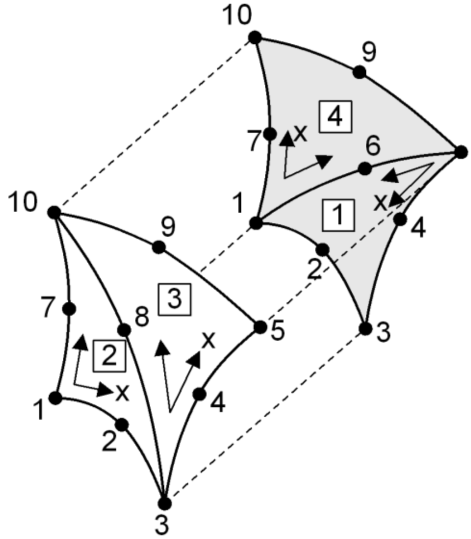
Note

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.

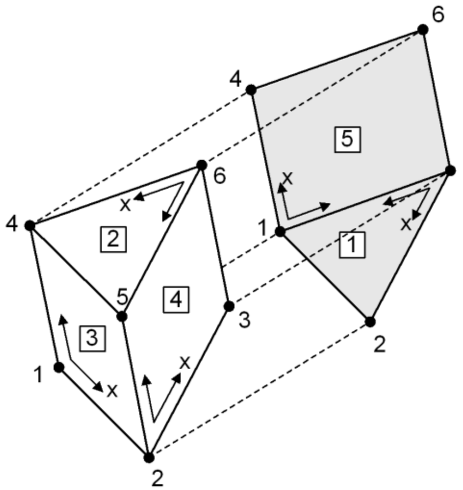
4-Noded Tetrahedra



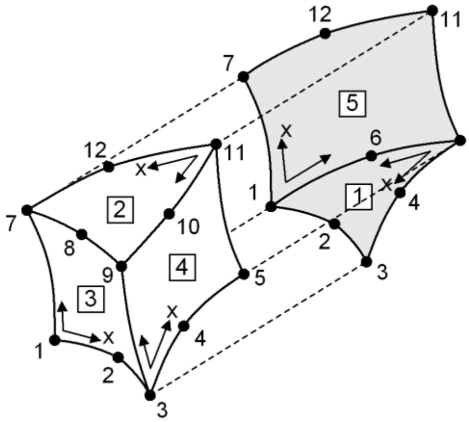
10-Noded Tetrahedra



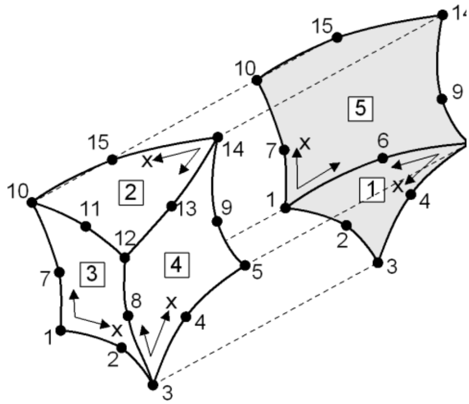
6-Noded Pentahedra



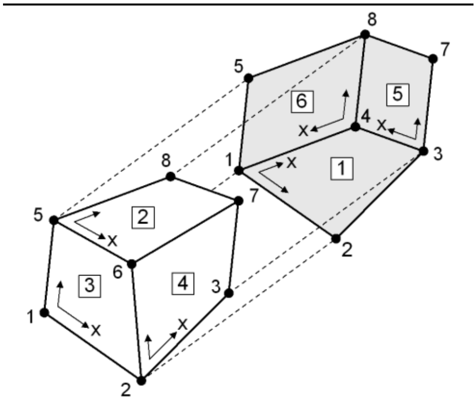
12-Noded Pentahedra



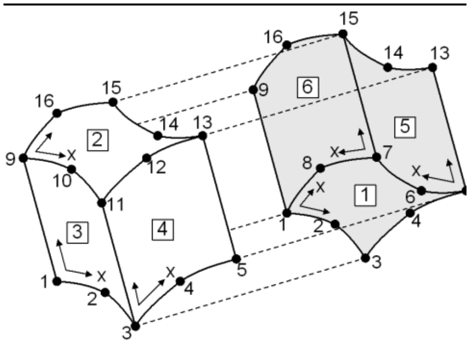
15-Node Pentahedra



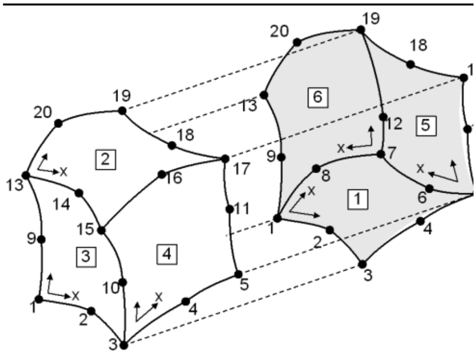
8-Noded Hexahedra



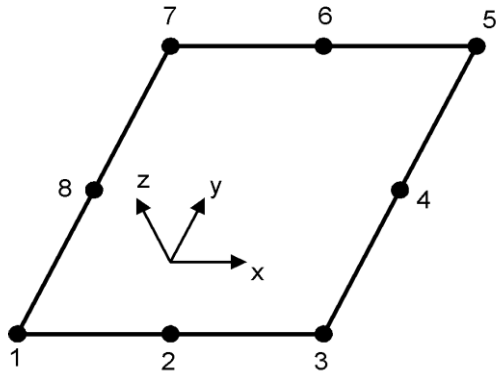
16-Noded Hexahedra



20-Noded Hexahedra



UDL Loads on Shells



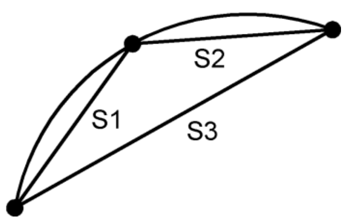
Appendix B : Element Restrictions.

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

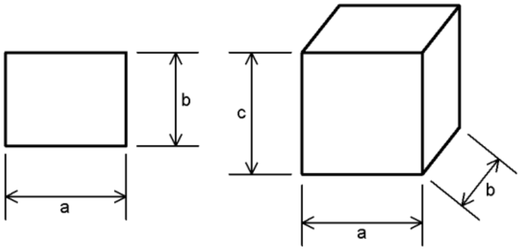
ii) (S1+S2) / S3 < 1.02

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, ...) 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

$$\mathbf{z < 0.01(L_{12})}$$

where **z** is the out of plane distance of a node,

and $\mathbf{L_{12}}$ 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.

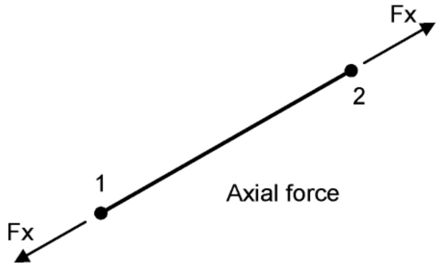
Appendix D : Sign Conventions.

The sign convention for forces, moments, stresses, rotations, eccentricities and potentials for different element types is documented in the following section headings.

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

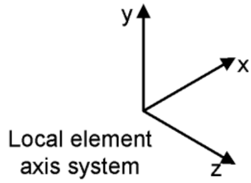
(+ve) Anti-clockwise rotation (1st node), clockwise rotation (3rd node)

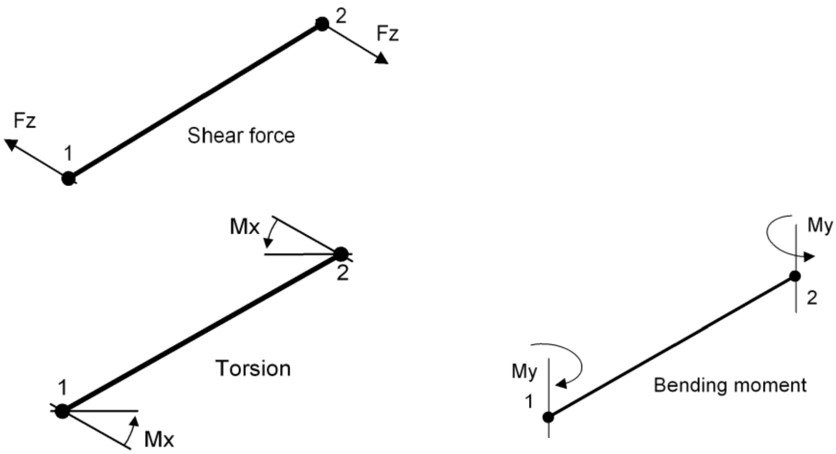
(-ve) Clockwise rotation (1st node), anti-clockwise rotation (3rd node)

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 My 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, Fz 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

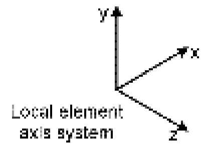
(-ve) Bottom of beam in tension

Where the top/bottom of the beam are determined by the element axes

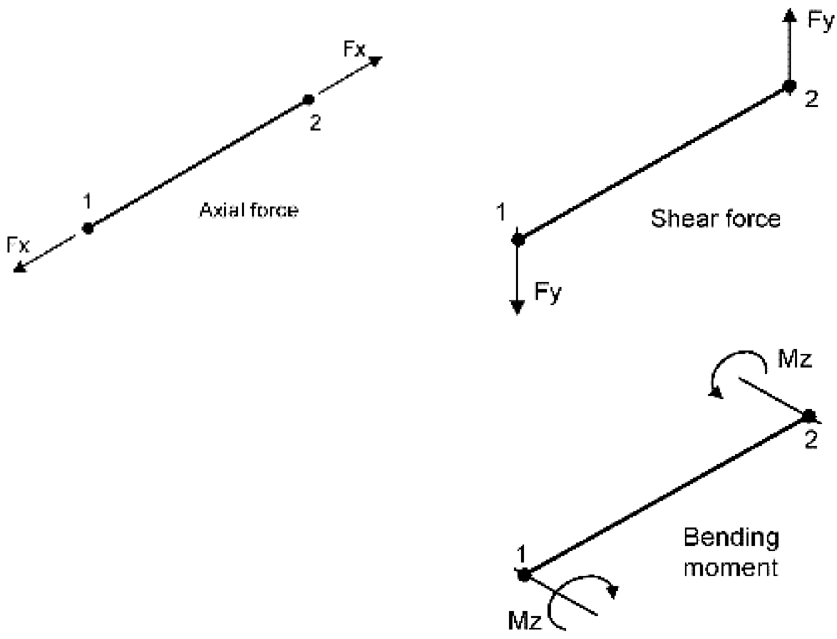
See **numerically integrated beam sign convention.**

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.

Axial force	Bending Moment	Torsion
(+ve) Axial tension (-ve) Axial compression	(+ve) Hogging moment (-ve) Sagging moment	(+ve) Anti-Clockwise rotation (end 1) (-ve) Clockwise rotation (end 2)

Sign convention in Modeller for bending moment

(+ve) Top of beam in tension

(-ve) Bottom of beam in tension

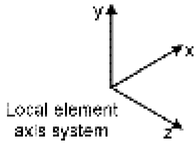
Where the top/bottom of the beam are determined by the element axes

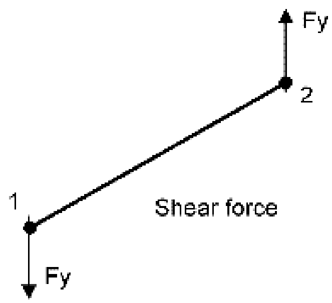
See **numerically integrated beam sign convention.**

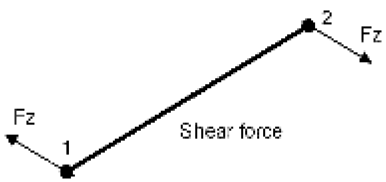
3D Engineering Beam Elements

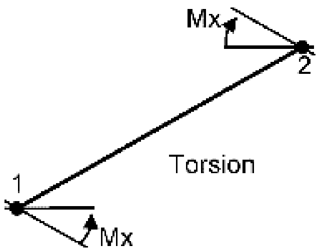
End Forces and Rotations

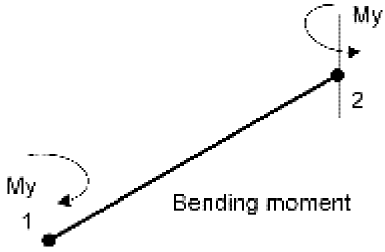
Positive end forces and rotations for 3D engineering beams are those acting on the element nodes in local directions, and are as follows:

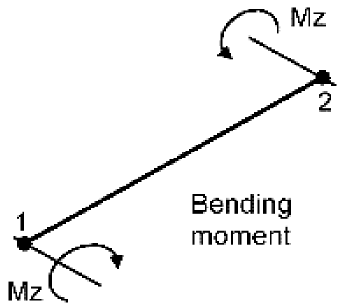












Internal forces

These forces follow the sign convention for numerically integrated beams.

Axial force	Bending Moment	Torsion
(+ve) Axial tension (-ve) Axial compression	(+ve) Hogging moment (-ve) Sagging moment	(+ve) Anti-Clockwise rotation (end 1) (-ve) Clockwise rotation (end 2)

Sign convention in Modeller for bending moment

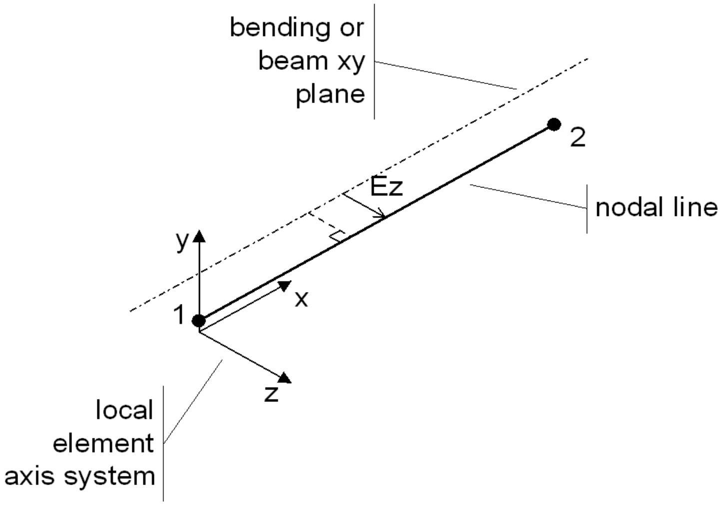
(+ve) Top of beam in tension

(-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

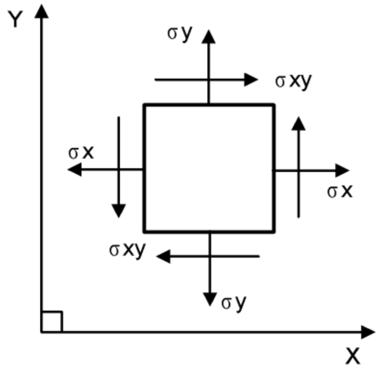


Eccentricities are optional geometric properties for some elements and may be specified if the nodal line of the element does not lie along the required bending line/plane for the structural component being modelled.

Measurement of Ez (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 Ey 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.

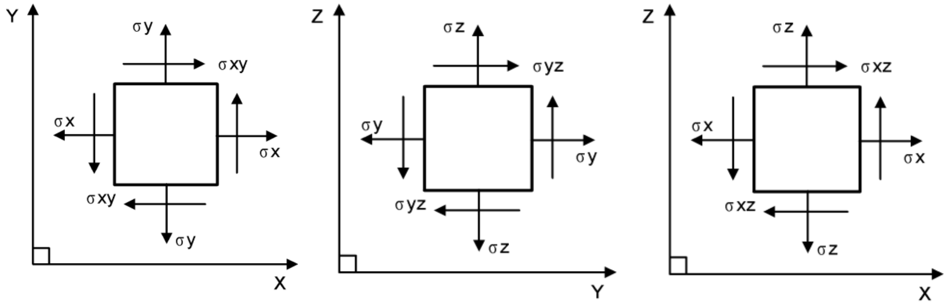
Standard 3D Continuum Element

Direct stress

(+ve) Tension
(-ve) Compression

Shear stress

(+ve) Shear into XY, YZ and XZ quadrants
(-ve) Shear into XY, YZ and XZ quadrants

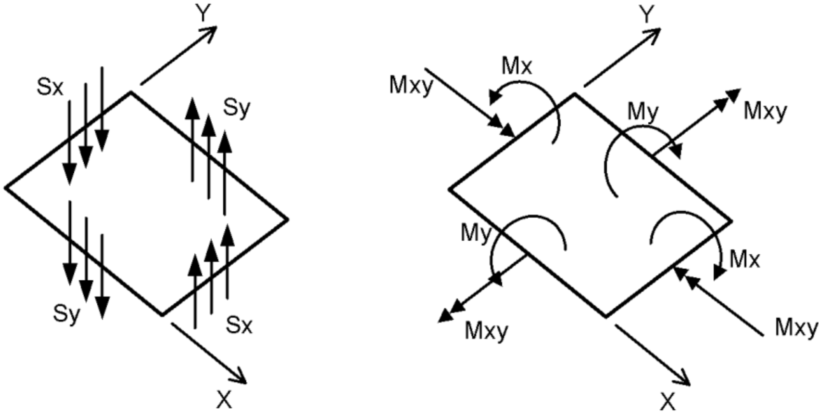


Note. Positive stress values shown.

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.

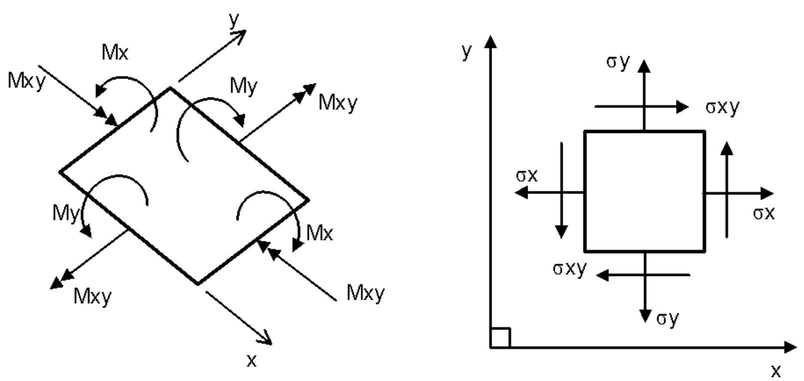
Thin Shell Element

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)

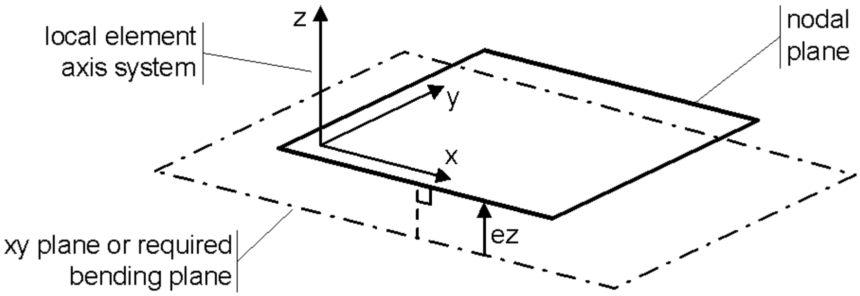


Notes

- Positive stress values 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.

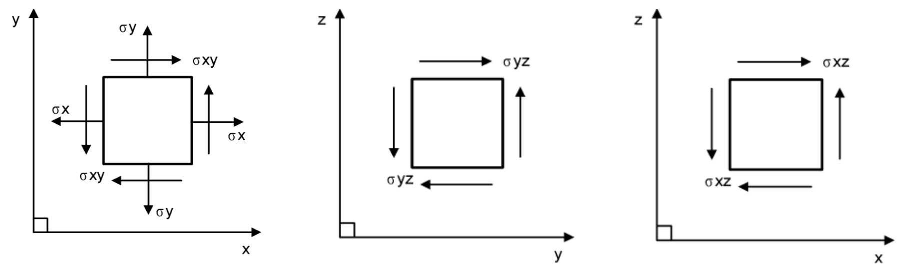


Measurement of e_z is **from** the required bending plane **to** the nodal plane in the local element axis z-direction.

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

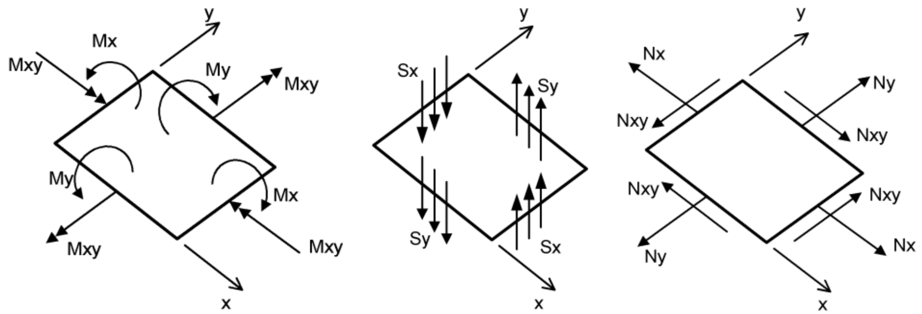


Stress Resultant

Membrane stress (+ve) Direct tension
(-ve) Direct compression

(+ve) In-plane shear into xy quadrant
(-ve) In-plane shear into xy quadrant

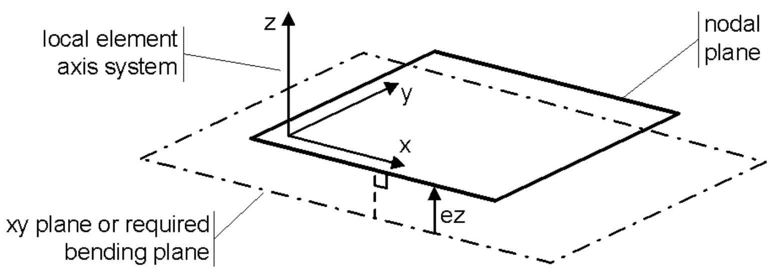
Flexural stress (+ve) Hogging moment (producing +ve stresses on the element top surface)
(-ve) Sagging moment (producing -ve stresses on the element top surface)



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.

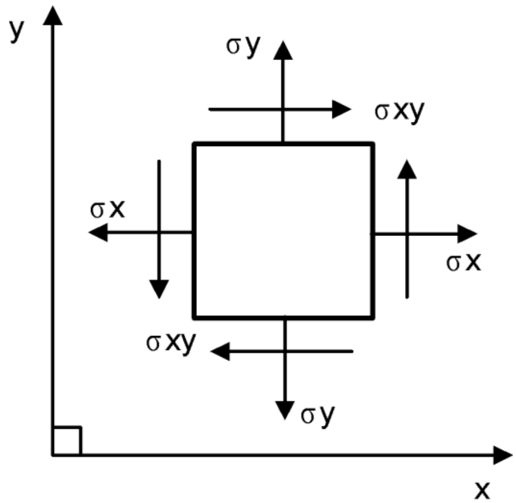


Measurement of ez is **from** the required bending plane **to** the nodal plane in the local element axis z-direction.

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.

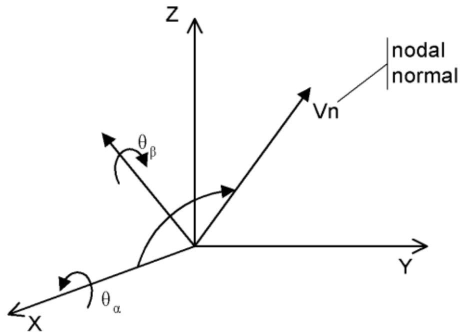
Thick Shell Nodal Rotation

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 θα 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 θβ. If the nodal normal coincides with the global Z axis, the global X axis will be chosen to define θα. 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 θα:

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, θX, θY and θZ 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.

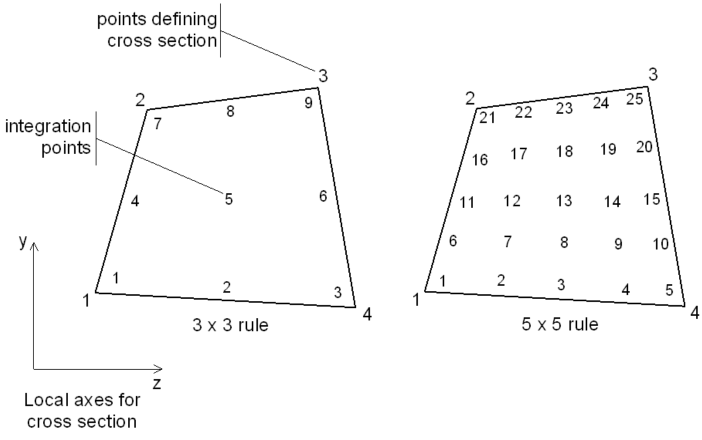
- When connecting with other element types. Six degrees of freedom will automatically be assigned to shell nodes connected to beams, joints or other shell element types.
- When boundary conditions or loading cannot be easily specified using the above definition of rotations, e.g. when applying moments or using symmetry.

If the rotations $\theta\alpha$, θ_β 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**.

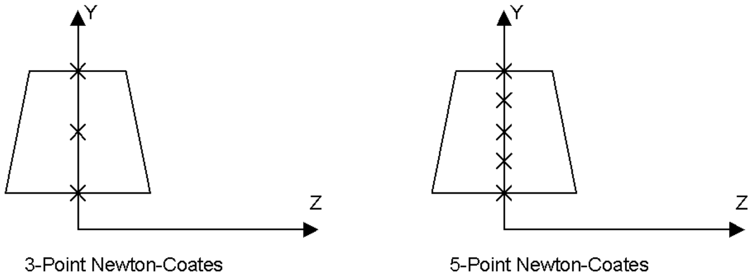
Appendix F : Newton Coates Integration.

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 (Asz, Asy). 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 Asz or Asy 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)

$$\frac{\pi . r^4}{2}$$

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 = $0.1406a^4$

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

$$\frac{2 \cdot t_1 \cdot t_2 \cdot (a - t_2)^2 (b - t_1)^2}{at_2 + bt_1 - t^2{}_2 - t^2{}_1}$$

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

$$\frac{1}{3}bt^3$$

where **b** is the rectangle length

and **t** is the rectangle length thickness

Any section consisting of thin rectangles

$$\frac{1}{3}\sum bt^3$$

Solid ellipse

$$\frac{\pi a^3 b^3}{a^2 + b^2}$$

where **2a** is the longest dimension

and **2b** is the shortest dimension

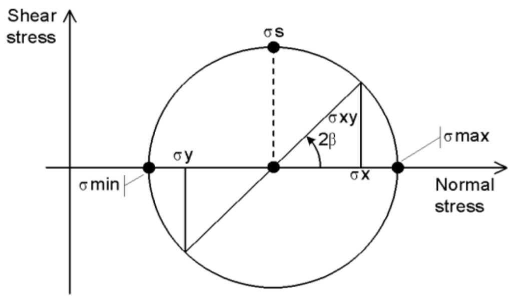
Note

- 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:

Smax is the maximum principal stress.

Smin is the minimum principal stress

Ss is the maximum shear stress

β defines the orientation of the principal axis (the plane on which the principal stresses act).

Sx, σy, σxy 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

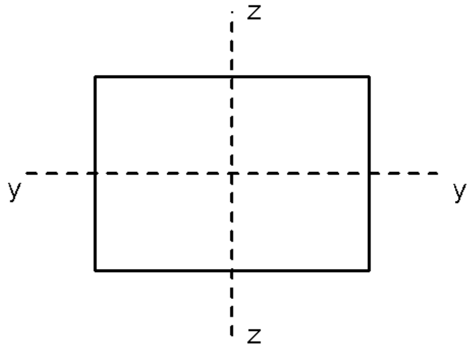
$$I_y = \int z dA$$

(=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

EZ Direct strain in Z direction

EXY Shear strain in XY plane

EYZ Shear strain in YZ plane

EZX Shear strain in XZ plane

EMax Maximum principal strain

EMin Minimum principal strain

E1 Major principal strain

E2 Intermediate principal strain

E3 Minor principal strain

Eabs Signed largest value of principal strain

axis

Bz Bending strain (curvature) about z axis

Bxy Bending or torsional strain into xy plane

Byz Bending or torsional strain into yz plane

Bxz Bending or torsional strain into xz plane

BMax Maximum principal bending strain

BMin Minimum principal bending strain

β Angle between E1 and X axis

EE Equivalent strain (von Mises)

EI Maximum shear strain

Strains: Top/Middle/Bottom (TMB)

EX Direct strain in X direction

EY Direct strain in Y direction

EZ Direct strain in Z direction

EXY Shear strain in XY plane

EYZ Shear strain in YZ plane

EXZ Shear strain in XZ plane

E1 Major principal strain

E2 Intermediate principal strain

E3 Minor principal strain

Eabs Signed largest value of principal strain

β Angle between E1 and X axis

EE Equivalent strain (von Mises)

EI Maximum shear strain

Plastic Strains

EPX Plastic direct strain in X direction

EPY Plastic direct strain in Y direction

EPZ Plastic direct strain in Z direction

EPXY Plastic shear strain in XY plane

EPYZ Plastic shear strain in YZ plane

EPZX Plastic shear strain in ZX plane

EP1 Major principal strain

EP2 Intermediate principal plastic strain

EP3 Minor principal plastic strain

EPabs Signed largest value of principal plastic strain

β Angle between EP1 and X axis

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 y
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
Nβετα	Angle between NMax and x axis	**Mβετα**	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	**Fc(B)**	Bottom surface concrete force

factor

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

MY(T) Top surface global Y moment

MX(B) Bottom surface global X moment

MY(B) Bottom surface global Y moment

MUtil(T) Top surface utilisation factor for bending only

MUtil(B) Bottom surface utilisation factor for bending only

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

FV (transverse) Force in Vertical direction for transverse members that are orthogonal or skewed in relation to the reference path

MF (longitudinal) Flexural Moment in longitudinal members that are following 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

Fluxes

qX Field flux in X direction

GY	Field gradient in Y direction	**qY**	Field flux in Y direction
GY	Field gradient in Z 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	**VFLW**	Velocity of Flow

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 + ··· + ni/Ni = 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 (StEngD)	**PWD**	Plastic work density

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.

Key to Slideline Results Components

This section contains the notation for slideline results. Note that slideline results components are not listed in the results tables.

TanGapFrcx	Tangential gap force in local x direction	**NrmPen**	Penetration normal to contact surface
TanGapFrcy	Tangential gap force in local y direction	**ContStatus**	In-contact/out-of-contact status
RsltTanGFc	Resultant tangential gap force	**ContacArea**	Nodal contact area
NrmGapForc	Gap force normal to contact surface	**Contact**	In-contact/out-of-contact status
ForceX	Contact force in system x direction	**Zone**	Zonal contact parameter
ForceY	Contact force in system y direction	**ZnCnDetDst**	Zonal contact detection distance
ForceZ	Contact force in system z direction	**IntStfCoef**	Contact stiffness coefficient
RsltForce	Resultant contact force	**TanForcex**	Tangential contact force in local x direction
ContStresx	Contact stress in local x direction	**TanForcey**	Tangential contact force in local y direction
ContStresy	Contact stress in local y	**RsltTanFrc**	Resultant tangential contact

	direction		force
ContPress	Contact pressure normal to contat surface	**NrmForce**	Contact force normal to contact surface
ContStiff	Contact stiffness		

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	DAMAM	DFUNC	SED	PWD
(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	StchI	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	ECI	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	StchI	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	StchI	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	StchI	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	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												
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 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	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												
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	StchI	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	StchI	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	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												
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.

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	MXY	MMax	MMin	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	EPMax	EMin	EI	β	Eabs	EE			
(continued)	DDAMA	CURRD	DAMAM	DFUNC	SED	PWD	Eadp				
TMB Plastic Strain	EPx	EPz	EPMax	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	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.

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	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	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	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	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.

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)

JL43	Semiloof Shells	TSL6, QSL8 (corner nodes)
JSH4	3D Beams	BMS3, BTS3, BS3, BS4, BSX4, BMI21, BMI31, BM122, 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, QAX8P, 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. If eccentricity is defined for a joint element (JPH3/JSH4/JL46), the joint will behave in the same manner as an infinitesimally short eccentric beam.
3. 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).
4. 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

3

A

B

O

P

Q

U