

OOFEM Element Library Manual

Bořek Patzák

Czech Technical University
Faculty of Civil Engineering
Department of Structural Mechanics
Thákurova 7, 166 29 Prague, Czech Republic

February 10, 2014

Contents

1	Introduction	3
2	Elements for Structural Analysis (SM Module)	3
2.1	Truss Elements	3
2.1.1	Truss 1D element	3
2.1.2	Truss 2D element	3
2.1.3	Truss 3D element	4
2.2	Beam Elements	5
2.2.1	Beam2d element	5
2.2.2	Beam3d element	6
2.3	LatticeElements	8
2.3.1	Lattice2d	8
2.4	Plane Stress Elements	9
2.4.1	PlaneStress2d	9
2.4.2	QPlaneStress2d	9
2.4.3	TrPlaneStress2d	10
2.4.4	QTrPlStr	10
2.4.5	TrPlaneStrRot	12
2.4.6	TrPlaneStressRotAllman	12
2.5	Plane Strain Elements	14
2.5.1	Quad1PlaneStrain	14
2.5.2	TrplaneStrain	14
2.6	Plate&Shell Elements	16
2.6.1	DKT Element	16
2.6.2	CCT Element	16
2.6.3	CCT3D Element	16
2.6.4	RerShell Element	17
2.6.5	tr_shell01 element	17
2.6.6	Quad1Mindlin Element	17
2.6.7	Tr2Shell7 Element	18
2.7	Axisymmetric Elements	22
2.7.1	Axisymm3d element	22
2.7.2	Q4axisymm element	23
2.7.3	L4axisymm element	23
2.8	3d Continuum Elements	25
2.8.1	LSpace element	25
2.8.2	LSpaceBB element	25
2.8.3	QSpace element	25
2.8.4	LTRSpace element	26
2.8.5	QTRSpace element	26
2.8.6	LWedge element	26
2.8.7	QWedge element	27
2.9	Interface elements	30
2.9.1	Interface1d element	30
2.9.2	Interface2dquad element	30
2.9.3	Interface3dtrlin element	30
2.10	XFEM elements	33
2.10.1	TrPlaneStress2dXFEM element	33
2.10.2	PlaneStress2dXfem element	33
2.11	Iso Geometric Analysis based (IGA) elements	35
2.12	Special elements	36
2.12.1	LumpedMass element	36
2.12.2	Spring element	36
2.13	Geometric nonlinear analysis	38

3	Elements for Transport problems (TM Module)	39
3.1	2D Elements	39
3.1.1	Quad1ht element	39
3.1.2	Quad1hmt element	39
3.1.3	Tr1ht element	40
3.2	Axisymmetric Elements	41
3.2.1	Quadaxisym1ht element	41
3.2.2	Traxisym1ht element	41
3.3	3D Elements	42
3.3.1	Tetrah1ht - tetrahedral 3D element	42
3.3.2	Brick1ht - hexahedral 3D element	42
3.3.3	Brick1hmt - hexahedral 3D element	43
3.3.4	QBrick1ht - quadratic hexahedral 3D element	43
3.3.5	QBrick1hmt - quadratic hexahedral 3D element	43
4	Elements for Fluid Dynamics problems (FM Module)	45
4.1	Stokes' Flow Elements	45
4.1.1	Tr21Stokes element	45
4.1.2	Tet21Stokes element	45
4.1.3	Hexa21Stokes element	45
4.1.4	Tr1BubbleStokes element	46
4.1.5	Tet1BubbleStokes element	46
4.2	2D CBS Elements	48
4.2.1	Tr1CBS element	48
4.3	2D SUPG/PSGP Elements	49
4.3.1	Tr1SUPG element	49
4.3.2	Tr21SUPG element	49
4.3.3	Tr1SUPGAxi element	50
4.4	3D SUPG/PSGP Elements	52
4.4.1	Tet1.3D.SUPG element	52

List of Figures

1	Truss2d element in (x,z) plane.	3
2	Beam2d element. Definition of local c.s.(a) and definition of local end forces and local element dofs (b).	5
3	Beam3d element. Definition of local c.s., local end forces and local element dofs numbering. . .	6
4	Lattice2d element. Node numbering, DOF numbering and definition of integration point C . . .	8
5	PlaneStress2d element. Node numbering, edge numbering and definition of local edge c.s.(a). .	9
6	QPlaneStress2d element - node numbering.	10
7	TrPlaneStress2d element - node and side numbering.	10
8	QTrPlStr element - node and side numbering.	11
9	Quad1PlaneStrain element. Node numbering, Side numbering and definition of local edge c.s.(a). .	14
10	TrplaneStrain element - node and side numbering.	15
11	Geometry of tr_shell01 element.	18
12	LSpace element (Node numbers in black, side numbers in blue, and surface numbers in red). . .	25
13	QSpace element.	26
14	LTRSpace element. Definition and node numbering convention.	27
15	QTRSpace element. Definition and node numbering convention.	27
16	LWedge element. Node numbering convention in black, edge numbering in blue and face numbering in red.	28
17	QWedge element. Node numbering convention in black, edge numbering in blue and face numbering in red.	28
18	Interface2dquad element with quadratic interpolation. Definition and node numbering convention	31
19	Interface3dtrlin element with linear interpolation. Definition and node numbering convention .	31
20	Quad1ht element. Node numbering, Side numbering and definition of local edge c.s.(a). . . .	39
21	Tr1ht element - node and side numbering.	40
22	Brick1ht element. Node numbers are in black, side numbers are in blue, and surface numbers are in red.	42
23	Hexa21Stokes element. Node numbering and face numbering.	46
24	Tr1CBS element. Node numbering, Side numbering and definition of local edge c.s.(a). . . .	48
25	Tr1SUPG element. Node numbering, Side numbering and definition of local edge c.s.(a). . . .	49
26	Tr21SUPG element - node and side numbering.	50
27	Tr1SUPGAXI element. Node numbering, Side numbering and definition of local edge c.s.(a). . .	51
28	Tet1.3D.SUPG element.	52

1 Introduction

In this manual the detailed description of available elements is given. The actual availability of particular elements depends on OOFEM configuration. Elements are specified using element records, which are part of oofem input file. The general format of element record is described in OOFEM input manual.

Every element is described in a separate section. The section includes the “element keyword”, which determines the element type in element record, approximation and interpolation characteristics, required cross section properties (which are summarized in “CS properties” part), and a summary of element features. The “Load” section contains useful information about the types of loadings supported by particular elements.

2 Elements for Structural Analysis (SM Module)

2.1 Truss Elements

2.1.1 Truss 1D element

Represents linear isoparametric truss element in 1D. The elements are assumed to be located along the x-axis. Requires cross section area to be specified. The element features are summarized in Table 1.

Keyword	truss1d
Description	1D truss element
Specific parameters	-
Unknowns	Single dof (u-displacement) is required in each node
Approximation	Linear approximation of displacement and geometry
Integration	Exact
Features	Full dynamic analysis support, Full nonlocal constitutive support, Adaptivity support
CS properties	Area is required
Loads	Body loads are supported. Boundary loads are not supported in current implementation
Status	Reliable

Table 1: truss1d element summary

2.1.2 Truss 2D element

Two node linear isoparametric truss element for 2D analysis. The element geometry can be specified in (x,z), (x,y), or (y,z) plane. The element features are summarized in Table 2.

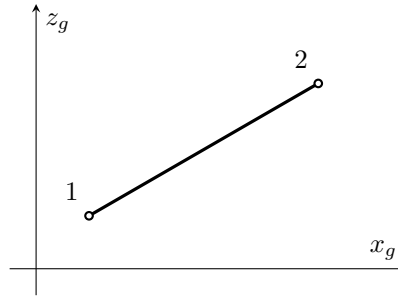


Figure 1: Truss2d element in (x,z) plane.

Keyword	truss2d
Description	2D truss element
Specific parameters	[cs # _(in)]
Parameters	cs : this parameter can be used to change default definition plane. The supported values of cs are following: 0 for (x,z) plane (default), 1 for (x,y) plane, and 3 for (y,z) plane.
Unknowns	Two dofs representing displacements in definition plane are required in each node. The element can be used in different planes, default definition plane is (x,z). The parameter cs can be used to change default definition plane.
Approximation	Linear approximation of displacements and geometry.
Integration	Exact.
Features	Full dynamic analysis support. Full nonlocal constitutive support.
CS properties	cross section area should be provided.
Loads	Edge loads are supported, Edge number should be equal to 1
Status	Reliable

Table 2: truss2d element summary

truss2d element summarytruss2dsummary

2.1.3 Truss 3D element

Two node linear isoparametric truss element for 3D analysis. The element geometry is specified in (x,y,z) plane. The element features are summarized in Table 3.

Keyword Description Specific parameters	truss3d 3D truss element -
Unknowns Approximation Integration Features CS properties Status	Three displacement DOFs (in x, y, and z directions) are required in each node. Linear approximation of displacements and geometry. Exact. Full dynamic analysis support. Full nonlocal constitutive support. cross section area should be provided. Reliable

Table 3: truss3d element summary

2.2 Beam Elements

2.2.1 Beam2d element

Beam element for 2D analysis, based on Timoshenko hypothesis. Structure should be defined in x,z plane. The internal condensation of arbitrary DOF is supported and is performed in local coordinate system. On output, the local end displacement and local end forces are printed. The element features are summarized in Table 4.

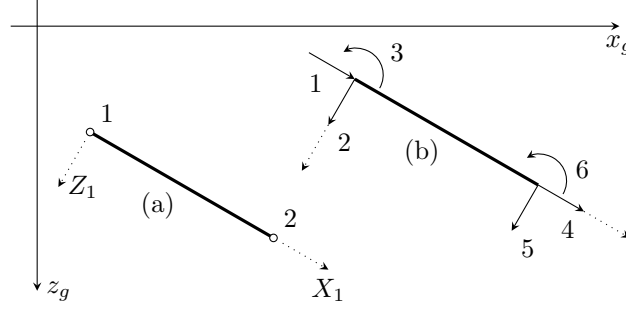


Figure 2: Beam2d element. Definition of local c.s.(a) and definition of local end forces and local element dofs (b).

Keyword	beam2d
Description	2D beam element
Specific parameters	[dofstocondense # _(ia)]
Parameters	dofstocondense: allows to specify local element dofs that will be condensed. The numbering of local element dofs is shown in fig. 2. The size of this array should be equal to number of local element dofs (6) and nonzero value indicates the corresponding dof will be condensed.
Unknowns	Three dofs (u-displacement, w-displacement, y-rotation) are required in each node.
Approximation	Cubic approximations of lateral displacement and rotation are used. For longitudinal displacement the linear one is assumed.
Integration	Exact.
Features	Full dynamic analysis support. Linear stability analysis support.
CS properties	Area, inertia moment along y-axis (iy parameter) and equivalent shear area (shearareaz parameter) should be specified.
Loads	Constant and linear edge loads are supported, shear influence is taken into account. Edge number should be equal to 1. Temperature load is supported, the first coefficient of temperature load represent mid-plane temperature change, the second one represent difference between temperature change of local z+ and local z- surfaces of beam (in local coordinate system). Temperature load require that the “thick” property of cross section model is defined.
Status	Reliable

Table 4: beam2d element summary

2.2.2 Beam3d element

Beam element for 3D **linear** analysis, based on Timoshenko hypothesis. The internal condensation of arbitrary DOF is supported and is performed in local coordinate system. On output, the local end-displacement and local end-forces are printed. Requires the local coordinate system to be chosen according to main central axes of inertia. Local element coordinate system is determined by the following rules:

1. let first element node has following coordinates (x_i, y_i, z_i) and the second one (x_j, y_j, z_j) ,
2. direction vector of local x-axis is then $\mathbf{a}_1 = (x_j - x_i, y_j - y_i, z_j - z_i)$,
3. local y-axis direction vector lies in plane defined by local x-axis direction vector ($\mathbf{a}_1$) and given point (k-node with coordinates (x_k, y_k, z_k)) - so called reference node,
4. local z-axis is then determined as vector product of local x-axis direction vector ($\mathbf{a}_1$) by vector $(x_k - x_i, y_k - y_i, z_k - z_i)$,
5. local y-axis is then determined as vector product of local z-axis direction vector by local x-axis direction vector.

The element features are summarized in Table 5.

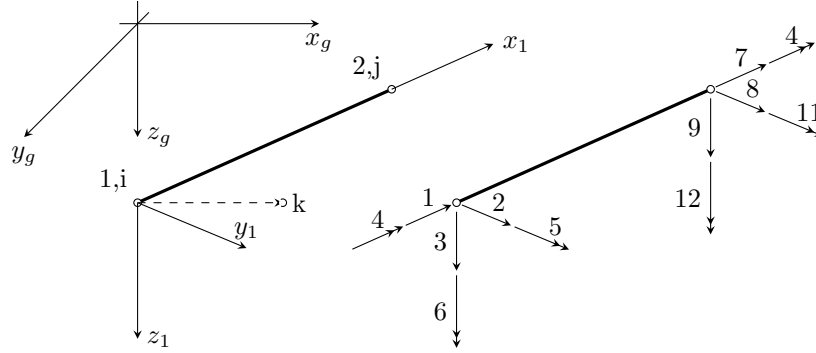


Figure 3: Beam3d element. Definition of local c.s., local end forces and local element dofs numbering.

Keyword	beam3d
Description	3D beam element
Specific parameters	refnode $\#_{(in)}$ [dofstocondense $\#_{(ia)}$]
Parameters	refnode : sets reference node to determine the local coordinate system of element. dofstocondense : allows to specify local element dofs that will be condensed. The numbering of local element dofs is shown in fig. 3. The size of this array should be equal to number of local element dofs (12) and nonzero value indicates the corresponding dof will be condensed.
Unknowns	Six dofs (u,v,w-displacements and x,y,z-rotations) are required in each node.
Approximation	Cubic approximations of lateral displacement and rotation (along local y,z axes) are used. For longitudinal displacement and the rotation along local x-axis (torsion) the linear approximations are assumed.
Integration	Exact.
Features	Full dynamic analysis support. Linear stability analysis support.
CS properties	Area, inertia moment along y and z axis (iy and iz parameters), torsion inertia moment (ik parameter) and either cross section area shear correction factor (beamshearccoeff parameter) or equivalent shear areas (shearareay and shearareaz parameters) are required. These cross section properties are assumed to be defined in local coordinate system of element.
Loads	Constant and linear edge loads are supported. Edge number should be equal to 1. Temperature load is supported, the first coefficient of temperature load represent mid-plane temperature change, the second one represent difference between temperature change of local z+ surface and local z-surface surface of beam and the third one represent difference between temperature change of local y+ surface and local y-surface of beam. Requires the “thick” (measured in direction of local z axis) and “width” (measured in direction of local y axis) cross section model properties to be defined.
Status	Reliable

Table 5: beam3d element summary

2.3 LatticeElements

2.3.1 Lattice2d

Represents two-node lattice element. Each node has 3 degrees of freedom. The element is defined in x,y plane. The element features are summarized in Table 6.

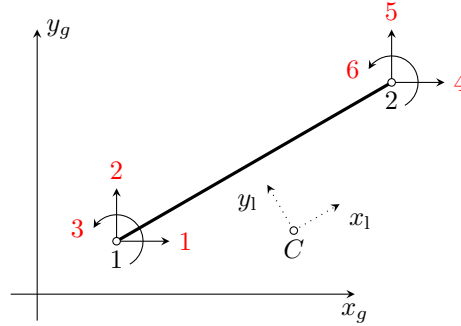


Figure 4: Lattice2d element. Node numbering, DOF numbering and definition of integration point C .

Keyword	lattice2d
Description	Lattice element
Specific parameters	thick $\#_{(rn)}$ width $\#_{(rn)}$ gpCoords $\#_{(ra)}$
Parameters	thick : defines the out of plane (z -direction) thickness width : defines the width of the midpoint cross-section in the x - y plane with the point C at its centre gpCoords : array of the coordinates of the integration point C in the global coordinate system
Unknowns	Three dofs (u -displacement, v -displacement, w -rotation) are required in each node.

Table 6: lattice2d element summary

2.4 Plane Stress Elements

2.4.1 PlaneStress2d

Represents isoparametric four-node quadrilateral plane-stress finite element. Each node has 2 degrees of freedom. Structure should be defined in x,y plane. The nodes should be numbered anti-clockwise (positive rotation around z -axis). The element features are summarized in Table 7.

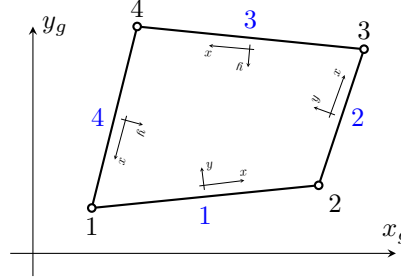


Figure 5: PlaneStress2d element. Node numbering, edge numbering and definition of local edge c.s.(a).

Keyword	planestress2d
Description	2D quadrilateral element for plane stress analysis
Specific parameters	[NIP # _(in)]
Parameters	NIP: allows to set the number of integration points
Unknowns	Two dofs (u-displacement, v-displacement) are required in each node.
Approximation	Linear approximation of displacements and geometry.
Integration	Integration of membrane strain terms using Gauss integration formula in 1, 4 (default), 9 or 16 integration points. The default number of integration points used can be overloaded using NIP parameter. Reduced integration for shear terms is employed. Shear terms are always integrated using the 1-point integration rule.
Features	Nonlocal constitutive support, Geometric nonlinearity support.
CS properties	cross section thickness is required.
Loads	Body loads are supported. Boundary loads are supported and computed using numerical integration. The side numbering is following. Each i -th element side begins in i -th element node and ends on next element node ($i+1$ -th node or 1-st node, in the case of side number 4). The local positive edge x -axis coincides with side direction, the positive local edge y -axis is rotated 90 degrees anti-clockwise (see fig. (5)).
Nlgeo	0, 1.
Status	Reliable

Table 7: planestress2d element summary

2.4.2 QPlaneStress2d

Implementation of quadratic isoparametric eight-node quadrilateral plane-stress finite element. Each node has 2 degrees of freedom. The node numbering is anti-clockwise and is explained in fig. (6). The element features are summarized in Table 8.

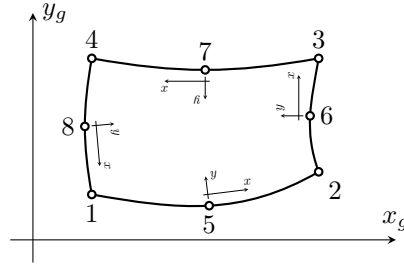


Figure 6: QPlaneStress2d element - node numbering.

Keyword	qplanestress2d
Description	2D quadratic isoparametric plane stress element
Specific parameters	[NIP # _(in)]
Parameters	NIP: allows to set the number of integration points
Unknowns	Two dofs (u-displacement, v-displacement) are required in each node.
Approximation	Quadratic approximation of displacements and geometry.
Integration	Full integration using Gauss integration formula in 4 (the default), 9 or 16 integration points. The default number of integration points used can be overloaded using NIP parameter.
Features	Adaptivity support.
CS properties	Cross section thickness is required.
Loads	Body and boundary loads are supported.
Nlgeo	0, 1.
Status	Stable

Table 8: qplanestress2d element summary

2.4.3 TrPlaneStress2d

Implements an triangular three-node constant strain plane-stress finite element. Each node has 2 degrees of freedom. The node numbering is anti-clockwise. The element features are summarized in Table 9.

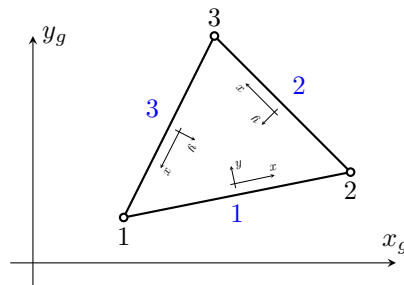


Figure 7: TrPlaneStress2d element - node and side numbering.

2.4.4 QTrPlStr

Implementation of quadratic six-node plane-stress finite element. Each node has 2 degrees of freedom. Node numbering is anti-clockwise and is shown in fig. (8). The element features are summarized in Table 10.

Keyword	trplanestress2d
Description	2D linear triangular isoparametric plane stress element
Specific parameters	-
Unknowns	Two dofs (u-displacement, v-displacement) are required in each node.
Approximation	Linear approximation of displacements and geometry.
Integration	Integration of membrane strain terms using one point gauss integration formula.
Features	Nonlocal constitutive support, Edge load support, Geometric nonlinearity support, Adaptivity support.
CS properties	Cross section thickness is required.
Loads	Body loads are supported. Boundary loads are supported and are computed using numerical integration. The side numbering is following. Each i-th element side begins in i-th element node and ends on next element node (i+1-th node or 1-st node, in the case of side number 3). The local positive edge x-axis coincides with side direction, the positive local edge y-axis is rotated 90 degrees anti-clockwise (see fig. (7)).
Nlgeo	0, 1.
Status	Reliable

Table 9: trplanestress2d element summary

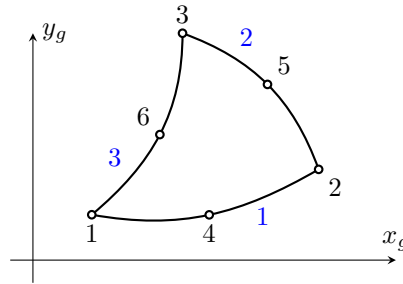


Figure 8: QTrPlStr element - node and side numbering.

Keyword	qtrplstr
Description	2D quadratic triangular plane stress element
Specific parameters	[NIP # _(in)]
Parameters	NIP: allows to set the number of integration points
Unknowns	Two dofs (u-displacement, v-displacement) are required in each node.
Approximation	Quadratic approximation of displacements and geometry.
Integration	Full integration using gauss integration formula in 4 points (the default) or in 7 points (using NIP parameter).
Features	Adaptivity support (error indicator).
CS properties	Cross section thickness is required.
Loads	Boundary loads are supported.
Nlgeo	0, 1.
Status	-

Table 10: qtrplstr element summary

2.4.5 TrPlaneStrRot

Implementation of triangular three-node plane-stress finite element with independent rotation field. Each node has 3 degrees of freedom. The element features are summarized in Table 11. The implementation is based on the following paper: Ibrahimbegovic, A., Taylor, R.L., Wilson, E. L.: A robust membrane quadrilateral element with rotational degrees of freedom, Int. J. Num. Meth. Engng., 30, 445-457, 1990. The rotation field is defined as $\omega = \frac{1}{2}(\frac{dv}{dx} - \frac{du}{dy}) = \nabla_u \mathbf{u}$. The following form of potential energy functional is assumed:

$$\Pi = \frac{1}{2} \int_{\Omega} \boldsymbol{\sigma}^T \boldsymbol{\varepsilon} d\Omega + \int_{\Omega} \boldsymbol{\tau}^T (\nabla_u \mathbf{u} - \omega) d\Omega - \int_{\Omega} \mathbf{X}^T \mathbf{u} d\Omega$$

where $\boldsymbol{\tau}$ is pseudo-stress (component of anti-symmetric stress tensor) working on dislocation ($\nabla_u \mathbf{u} - \omega$); the following constitutive relation foris assumed: $\boldsymbol{\tau} = G(\nabla_u \mathbf{u} - \omega)$, where G is elasticity modulus in shear.

Keyword	trplanestrrot
Description	2D linear triangular plane stress element with rotational DOFs
Specific parameters	[NIP # _(in)] [NIPRot # _(in)]
Parameters	NIP: allows to set the number of integration points for integration of membrane terms. NIPRot: allows to set the number of integration points for integration of terms associated to rotational field.
Unknowns	Three dofs (u-displacement, v-displacement, z-rotation) are required in each node.
Approximation	Linear approximation of displacements and geometry.
Integration	Integration of membrane strain terms using gauss integration formula in 4 points (default) or using 1 or 7 points (using NIP parameter). Integration of strains associated with rotational field integration using 1 point is default (4 and 7 points rules can be specified using NIPRot parameter).
Features	-
CS properties	- Cross section thickness is required.
Loads	-
Nlgeo	0.
Status	-

Table 11: trplanestrrot element summary

2.4.6 TrPlaneStressRotAllman

Implementation of triangular three-node plane-stress with nodal rotations. Each node has 3 degrees of freedom. The element features are summarized in Table 12. The implementation is based on the following paper: Allman, D.J.: A compatible triangular element including vertex rotations for plane elasticity analysis, Computers & Structures, vol. 19, no. 1-2, pp. 1-8, 1984. The element is based on plane stress element with quadratic interpolation. The displacements in midside nodes are expressed using vertex displacements and vertex rotations (for edge normal displacement component); the tangential component is interpolated from vertex values. For particular element side starting at i-th vertex and ending in j-th vertex the normal and tangential displacements at edge midpoint can be expressed as

$$\begin{aligned} u_n|_{l/2} &= \frac{u_{ni} + u_{nj}}{2} + \frac{l}{8}(\omega_i - \omega_j) \\ u_t|_{l/2} &= \frac{u_{ti} + u_{tj}}{2} \end{aligned}$$

where l is edge length. This allows to express global displacements in element midside nodes using vertex displacements and rotations. For a single edge, one obtains:

$$\begin{aligned} u|_{l/2} &= -\frac{u_{ni} + u_{nj}}{2} + \frac{l}{8}(\omega_i - \omega_j)\frac{\Delta y_{ji}}{l} + \left(\frac{u_{t1} + u_{t2}}{2}\right)\frac{\Delta x_{ji}}{l} \\ v|_{l/2} &= \frac{u_{ni} + u_{nj}}{2} + \frac{l}{8}(\omega_i - \omega_j)\frac{\Delta x_{ji}}{l} + \left(\frac{u_{t1} + u_{t2}}{2}\right)\frac{\Delta y_{ji}}{l} \end{aligned}$$

Keyword	trplanestressrotallman
Description	2D linear triangular plane stress element with rotational DOFs
Specific parameters	
Unknowns	Three dofs (u-displacement, v-displacement, z-rotation) are required in each node.
Approximation	Linear approximation of geometry, quadratic interpolation of displacements.
Integration	Integration of membrane strain terms using gauss integration formula in 4 points.
Zero energy mode	The zero energy mode (equal rotations) is handled by adding additional energy term preventing spurious modes.
Features	-
CS properties	Cross section thickness is required.
Loads	-
Nlgeo	0.
Status	-

Table 12: trplanestressrotallman element summary

2.5 Plane Strain Elements

2.5.1 Quad1PlaneStrain

Represents isoparametric four-node quadrilateral plane-strain finite element. Each node has 2 degrees of freedom. Structure should be defined in x,y plane. The nodes should be numbered anti-clockwise (positive rotation around z -axis). The element features are summarized in Table 13.

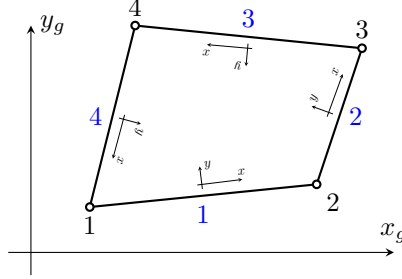


Figure 9: Quad1PlaneStrain element. Node numbering, Side numbering and definition of local edge c.s.(a).

Keyword	quad1planestrain
Description	2D linear quadrilateral plane-strain element
Specific parameters	[NIP # _(in)]
Parameters	NIP: allows to set the number of integration points for integration of membrane terms.
Unknowns	Two dofs (u-displacement, v-displacement) are required in each node.
Approximation	Linear approximation of displacements and geometry.
Integration	Integration of membrane strain terms using gauss integration formula in 4 (the default), 9 or 16 integration points. The default number of integration points used can be overloaded using NIP parameter. Reduced integration for shear terms is employed. Shear terms are always integrated using 1 point integration rule.
Features	Nonlocal constitutive support, Adaptivity support.
CS properties	Cross section thickness is required.
Loads	Body loads are supported. Boundary loads are supported and computed using numerical integration. The side numbering is following. Each i -th element side begins in i -th element node and ends on next element node ($i+1$ -th node or 1-st node, in the case of side number 4). The local positive edge x -axis coincides with side direction, the positive local edge y -axis is rotated 90 degrees anti-clockwise (see fig. (9)).
Nlgeo	0.
Status	Reliable

Table 13: quad1planestrain element summary

2.5.2 TrplaneStrain

Implements an triangular three-node constant strain plane-strain finite element. Each node has 2 degrees of freedom. The node numbering is anti-clockwise. The element features are summarized in Table 14.

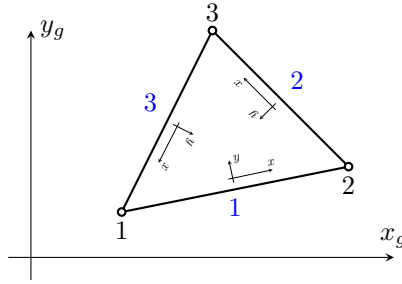


Figure 10: TrplaneStrain element - node and side numbering.

Keyword	trplanestrain
Description	2D linear triangular plane-strain element
Specific parameters	-
Unknowns	Two dofs (u-displacement, v-displacement) are required in each node.
Approximation	Linear approximation of displacements and geometry.
Integration	Integration of membrane strain terms using one point gauss integration formula.
Features	Nonlocal constitutive support. Edge load support, Adaptivity support.
CS properties	Cross section thickness is required.
Loads	Body loads are supported. Boundary loads are supported and are computed using numerical integration. The side numbering is following. Each i-th element side begins in i-th element node and ends on next element node (i+1-th node or 1-st node, in the case of side number 3). The local positive edge x-axis coincides with side direction, the positive local edge y-axis is rotated 90 degrees anti-clockwise (see fig. (10)).
Nlgeo	0.
Status	Reliable

Table 14: trplanestrain element summary

2.6 Plate&Shell Elements

2.6.1 DKT Element

Implementation of Discrete Kirchhoff Triangle (DKT) plate element. This element is suitable for thin plates, as the transverse shear strain energy is neglected. The structure should be defined in x,y plane, nodes should be numbered anti-clockwise (positive rotation around z-axis). The element features are summarized in Table 15.

Keyword	dktplate
Description	2D Discrete Kirchhoff Triangular plate element
Specific parameters	-
Unknowns	Three dofs (w-displacement, u and v - rotations) are required in each node.
Approximation	Quadratic approximation of rotations, cubic approximation of displacement along the edges. Note: there is no need to define interpolation for displacement on the element.
Integration	Default integration of all terms using three point formula.
Features	Layered cross section support.
CS properties	Cross section thickness is required.
Loads	Body loads are supported. Boundary load support is beta.
Output	On output, the generalized strains are printed in a vector with 8 components, with the following meaning: $e = \{\varepsilon_x, \varepsilon_y, \gamma_{xy}, \kappa_x, \kappa_y, \kappa_{xy}, \gamma_{zx}, \gamma_{zy}\},$ where $\varepsilon_x, \varepsilon_y, \varepsilon_z$ are membrane normal deformations, $\gamma_{zy}, \gamma_{zx}, \gamma_{xy}$ are (out of plane and in plane) shear components, and $\kappa_x, \kappa_y, \kappa_{xy}$ are curvatures. The generalized stress vector contains corresponding integral forces/moments: $s = \{n_x, n_y, n_{xy}, m_x, m_y, m_{xy}, v_x, v_y\}.$ Please note, for example, that bending moment m_x is defined as $m_x = \int \sigma_x z dz$, so it acts along the y-axis and positive value causes tension in bottom layer.
Nlgeo	0.
Status	Reliable

Table 15: dktplate element summary

2.6.2 CCT Element

Implementation of constant curvature triangular element for plate analysis. Formulation based on Mindlin hypothesis. The structure should be defined in x,y plane. The nodes should be numbered anti-clockwise (positive rotation around z-axis). The element features are summarized in Table 16.

2.6.3 CCT3D Element

Implementation of constant curvature triangular element for plate analysis. Formulation based on Mindlin hypothesis. The element could be arbitrarily oriented in space. The nodes should be numbered anti-clockwise (positive rotation around element normal). The element features are summarized in Table 17.

Keyword	cctplate
Description	2D constant curvature triangular plate element
Specific parameters	-
Unknowns	Three dofs (w-displacement, u and v - rotations) are required in each node.
Approximation	Linear approximation of rotations, quadratic approximation of displacement.
Integration	Integration of all terms using one point formula.
Features	Layered cross section support.
CS properties	Cross section thickness is required.
Loads	Body loads are supported. Boundary loads are not supported now.
Output	On output, the generalized strains are printed in a vector with 12 components, with the following meaning: $e = \{\varepsilon_x, \varepsilon_y, \varepsilon_z, \gamma_{yz}, \gamma_{zx}, \gamma_{xy}, \kappa_x, \kappa_y, \kappa_z, \kappa_{yz}, \kappa_{xz}, \kappa_{xy}\},$ where $\varepsilon_x, \varepsilon_y, \varepsilon_z$ are membrane normal deformations, $\gamma_{yz}, \gamma_{zx}, \gamma_{xy}$ are shear componets, and $\kappa_x, \kappa_y, \kappa_z, \kappa_{yz}, \kappa_{xz}, \kappa_{xy}$ are curvatures. The generalized stress vector contains corresponding integral forces/moments: $s = \{n_x, n_y, n_y, v_{yz}, v_{zx}, v_{xy}, m_x, m_y, m_z, m_{yz}, m_{xz}, m_{xy}\}.$ Please note, for example, that bending moment m_x is defined as $m_x = \int \sigma_x z dz$, so it acts along the y-axis and positive value causes tension in bottom layer.
Nlgeo	0.
Status	Reliable

Table 16: cctplate element summary

2.6.4 RerShell Element

Combination of CCT plate element (Mindlin hypothesis) with triangular plane stress element for membrane behavior. The element curvature can be specified. Although element requires generally six DOFs per node, no stiffness to local rotation along z-axis (rotation around element normal) is supplied. The element features are summarized in Table 18.

2.6.5 tr_shell01 element

Combination of CCT3D plate element (Mindlin hypothesis) with triangular plane stress element for membrane behavior. It comes with complete set of 6 DOFs per node. The element features are summarized in Table 19.

2.6.6 Quad1Mindlin Element

This class implements an quadrilateral, bilinear, four-node Mindlin plate. This type of element exhibit strong shear locking (thin plates exhibit almost no bending). Implements the lumped mass matrix. The element features are summarized in Table 21.

Keyword	cctplate3d
Description	Constant curvature triangular plate element in arbitray position
Specific parameters	-
Unknowns	Six dofs (u,v,w-displacements and u,v,w rotations) are in general required in each node.
Approximation	Linear approximation of ratations, quadratic approximation of displacement.
Integration	Integration of all terms using one point formula.
Features	Layered cross section support.
CS properties	Cross section thickness is required.
Loads	Body loads are supported. Boundary loads are not supported now.
Output	On output, the generalized strains are printed in a vector with 12 components, with the following meaning: $e = \{\varepsilon_x, \varepsilon_y, \varepsilon_z, \gamma_{yz}, \gamma_{zx}, \gamma_{xy}, \kappa_x, \kappa_y, \kappa_z, \kappa_{yz}, \kappa_{xz}, \kappa_{xy}\},$ where $\varepsilon_x, \varepsilon_y, \varepsilon_z$ are membrane normal deformations, $\gamma_{yz}, \gamma_{zx}, \gamma_{xy}$ are shear componets, and $\kappa_x, \kappa_y, \kappa_z, \kappa_{yz}, \kappa_{xz}, \kappa_{xy}$ are curvatures. The generalized stress vector contains corresponding integral forces/moments: $s = \{n_x, n_y, n_y, v_{yz}, v_{zx}, v_{xy}, m_x, m_y, m_z, m_{yz}, m_{xz}, m_{xy}\}.$ Please note, for example, that bending moment m_x is defined as $m_x = \int \sigma_x z dz$, so it acts along the y-axis and positive value causes tension in bottom layer.
Nlgeo	0.
Status	Reliable

Table 17: cctplate3d element summary

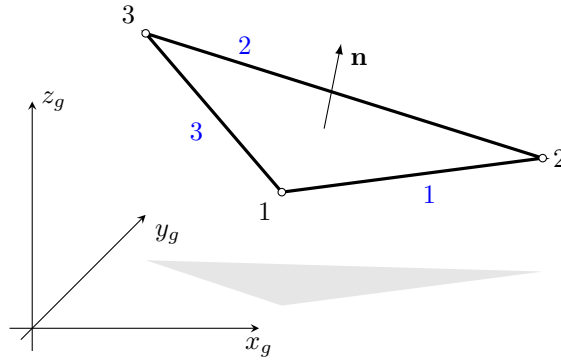


Figure 11: Geometry of tr_shell01 element.

2.6.7 Tr2Shell7 Element

This class implements a triangular, quadratic, six-node shell element. The element is a so-called seven parameter shell with seven dofs per node – a displacement field (3 dofs), an extensible director field (3 dofs) and a seventh dof representing inhomogenous thickness strain. This last parameter is included in the model in order to deal with volumetric/Poisson lock effects.

The element features are summarized in Table 21.

Keyword	rershell
Description	Simple shell based on combination of CCT plate element (Mindlin hypothesis) with triangular plane stress element. element can be arbitrary positioned in space.
Specific parameters	-
Unknowns	Six dofs (u,v,w-displacements and u,v,w rotations) are in general required in each node. Note, that although element it requires generally six DOFs per node, no stiffness to local rotation along z-axis (rotation around element normal) is supplied.
Approximation	Linear approximation of rotations, quadratic approximation of displacement.
Integration	Integration of all terms using one point formula.
Features	Layered cross section support.
CS properties	Cross section thickness is required.
Loads	Body loads are supported. Boundary loads are not supported now.
Output	On output, the generalized strains are printed in a vector with 12 components, with the following meaning: $e = \{\varepsilon_x, \varepsilon_y, \varepsilon_z, \gamma_{yz}, \gamma_{zx}, \gamma_{xy}, \kappa_x, \kappa_y, \kappa_z, \kappa_{yz}, \kappa_{xz}, \kappa_{xy}\},$ where $\varepsilon_x, \varepsilon_y, \varepsilon_z$ are membrane normal deformations, $\gamma_{yz}, \gamma_{zx}, \gamma_{xy}$ are shear componets, and $\kappa_x, \kappa_y, \kappa_z, \kappa_{yz}, \kappa_{xz}, \kappa_{xy}$ are curvatures. The generalized stress vector contains corresponding integral forces/moments: $s = \{n_x, n_y, n_y, v_{yz}, v_{zx}, v_{xy}, m_x, m_y, m_z, m_{yz}, m_{xz}, m_{xy}\}.$ Please note, for example, that bending moment m_x is defined as $m_x = \int \sigma_x z \, dz$, so it acts along the y-axis and positive value causes tension in bottom layer.
Nlgeo	0.
Status	Reliable

Table 18: rershell element summary

Keyword	tr_shell01
Description	Triangular shell element combining CCT3D plate element (Mindlin hypothesis) with triangular plane stress element with rotational DOFs
Specific parameters	-
Unknowns	Six dofs (u,v,w-displacements and u,v,w rotations) are in general required in each node.
Approximation	See description of cct and trplanstrrot elements
Integration	Integration of all terms using one point formula.
Features	Layered cross section support.
CS properties	Cross section thickness is required.
Loads	Body loads are supported. Boundary loads are not supported now.
Output	On output, the generalized strains are printed in a vector with 12 components, with the following meaning: $e = \{\varepsilon_x, \varepsilon_y, \varepsilon_z, \gamma_{yz}, \gamma_{zx}, \gamma_{xy}, \kappa_x, \kappa_y, \kappa_z, \kappa_{yz}, \kappa_{zx}, \kappa_{xy}\},$ where $\varepsilon_x, \varepsilon_y, \varepsilon_z$ are membrane normal deformations, $\gamma_{yz}, \gamma_{zx}, \gamma_{xy}$ are shear componets, and $\kappa_x, \kappa_y, \kappa_z, \kappa_{yz}, \kappa_{zx}, \kappa_{xy}$ are curvatures. The generalized stress vector contains corresponding integral forces/moments: $s = \{n_x, n_y, n_z, v_{yz}, v_{zx}, v_{xy}, m_x, m_y, m_z, m_{yz}, m_{zx}, m_{xy}\}.$ Please note, for example, that bending moment m_x is defined as $m_x = \int \sigma_x z \, dz$, so it acts along the y-axis and positive value causes tension in bottom layer.
Nlgeo	0.
Status	Reliable

Table 19: tr_shell01 element summary

Keyword	quad1mindlin
Description	Quadrilateral, bilinear, four-node Mindlin plate
Specific parameters	[NIP # _(in)]
Unknowns	Three dofs (w-displacement, u and v - rotation) are required in each node.
Approximation	Linear for all unknowns.
Integration	Default uses 4 integration points. No reduced integration is used, as it causes numerical problems.
Features	Layered cross section support.
CS properties	Cross section thickness is required.
Loads	Dead weight loads, and edge loads are supported.
Nlgeo	0.
Reference	[RobertCook1989]
Status	Experimental

Table 20: quad1mindlin element summary

Keyword	tr2shell7
Description	Triangular, quadratic, six-node shell with 7 dofs/node
Specific parameters	[NIP # _(in)]
Unknowns	Seven dofs (displacement in u, v and w-direction; change in director field in u, v and w-direction; and inhomogenous thickness stretch) are required in each node.
Approximation	Quadratic for all unknowns.
Integration	Default uses 6 integration points in the midsurface plane. Number of integration points in the thickness direction is determined by the Layered cross section.
Features	Layered cross section support.
CS properties	This element must be used with a Layered cross section.
Loads	Edge loads, constant pressure loads and surface loads are supported.
Nlgeo	Not applicable. The implementation is for large defomra-tions and hence geometrical nonlinearities will always be present, regardless the value of Nlgeo.
Reference	[RagnarLarsson2011]
Status	Experimental

Table 21: tr2shell7 element summary

2.7 Axisymmetric Elements

Implementation relies on elements located exclusively in x, y plane. The coordinate x corresponds to radius, y is the axis of rotation. Approximation of displacement functions u, v is carried out on a particular finite element. Nonzero strains read

$$\varepsilon_x = \varepsilon_r = \frac{\partial u}{\partial x} \quad (1)$$

$$\varepsilon_y = \varepsilon_z = \frac{\partial v}{\partial y} \quad (2)$$

$$\varepsilon_\theta = \frac{u}{r} \quad (3)$$

$$\gamma_{xy} = \gamma_{rz} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \quad (4)$$

Stress components can be computed from elasticity matrix. Note that this matrix corresponds to a submatrix of the full 3D elasticity matrix.

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_\theta \\ \sigma_{xy} \end{Bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 \\ \nu & 1-\nu & \nu & 0 \\ \nu & \nu & 1-\nu & 0 \\ 0 & 0 & 0 & (1-2\nu)/2 \end{bmatrix} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_\theta \\ \gamma_{xy} \end{Bmatrix} \quad (5)$$

In OOFEM, the strain vector is arranged as $\{\varepsilon_x, \varepsilon_y, \varepsilon_\theta, 0, 0, \gamma_{xy}\}^T$ and the stress vector $\{\sigma_x, \sigma_y, \sigma_\theta, 0, 0, \tau_{xy}\}^T$. Implementation assumes a segment of 1 rad.

2.7.1 Axisymm3d element

Implementation of triangular three-node finite element for axisymmetric continuum. Each node has 2 degrees of freedom. Node numbering and edge position is the same as in Fig. 7. The element features are summarized in Table 22.

Keyword	Axisymm3d
Description	Triangular axisymmetric linear element
Specific parameters	[NIP # _(in)] [NIPfish # _(in)]
Parameters	NIP: allows to set the number of integration points for integration of terms corresponding to ε_x and ε_y strains (possible completions are 1 (default), 4 and 7 point integration rule). NIPfish: allows to set the number of integration points for integration of remain terms (corresponding to ε_θ and γ_{rz}) (Supported values include 1 (default)).
Unknowns	Two dofs (u-displacement, v-displacement) are required in each node.
Approximation	Linear approximation of displacement and geometry.
Integration	The integration of ε_x and ε_y strains can be altered using NIP paramter (default is 1 point integration). The remaining terms corresponding to strain components ε_θ and γ_{rz} are by default evaluated at 1 integration point, this can be changed by NIPfish parameter.
Features	-
CS properties	-
Loads	Boundary and body loads are supported.
Nlgeo	0.
Status	-

Table 22: Axisymm3d element summary

2.7.2 Q4axisymm element

Implementation of quadratic isoparametric eight-node quadrilateral - finite element for axisymmetric 3d continuum. Each node has 2 degrees of freedom. The element features are summarized in Table 23.

Keyword	Q4axisymm
Description	Quadratic isoparametric eight-node quadrilateral for axisymmetric analysis
Specific parameters	[NIP # _(in)] [NIPfish # _(in)]
Parameters	NIP: allows to set the number of integration points for integration of terms corresponding to ε_x and ε_y strains (possible completions are 1, 4 (default), 9, and 16). NIPfish: allows to set the number of integration points for integration of remain terms (corresponding to ε_θ and γ_{rz}) (Supported values include 1 (default), 4, 9, and 16 integration point formula).
Unknowns	Two dofs (u-displacement, v-displacement) are required in each node.
Approximation	Quadratic approximation of displacement and geometry.
Integration	The integration of terms corresponding to ε_x and ε_y strains can be altered using NIP parameter (default is 4 point formula). The remaining terms (corresponding to ε_θ and γ_{rz}) are integrated by default using 1 point formula (see NIPfish parameter).
Features	-
CS properties	-
Loads	No boundary and body loads are supported.
Nlgeo	0.
Status	-

Table 23: Q4axisymm element summary

2.7.3 L4axisymm element

Implementation of isoparametric four-node quadrilateral axisymmetric finite element with linear interpolations of displacements u, v . Node numbering and edge position is the same as in Fig. 5. The element features are summarized in Table 24.

Keyword	L4axisymm
Description	Isoparametric four-node quadrilateral element for axisymmetric analysis
Specific parameters	[NIP # _(in)]
Parameters	NIP: allows to set the number of integration points for integration of terms corresponding to ε_x and ε_y strains (possible completions are 1, 4 (default), 9, and 16).
Unknowns	Two dofs (u-displacement, v-displacement) are required in each node.
Approximation	Linear approximation of displacement and geometry.
Integration	The integration of ε_x and ε_y strains can be altered using NIP parameter (possible completions are 1, 4 (default), 9 or 16 point integration rule). The remaining strain components (ε_θ and γ_{rz}) are integrated using one point integration formula.
Features	-
CS properties	-
Loads	Boundary and body loads supported.
Nlgeo	0.
Status	-

Table 24: L4axisymm element summary

2.8 3d Continuum Elements

2.8.1 LSpace element

Implementation of Linear 3d eight - node finite element. Each node has 3 degrees of freedom. The element features are summarized in Table 25.

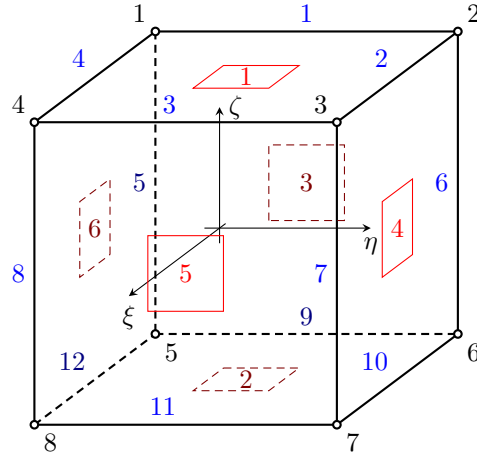


Figure 12: LSpace element (Node numbers in black, side numbers in blue, and surface numbers in red).

Keyword	lspace
Description	Linear isoparametric brick element
Specific parameters	[NIP # _(in)]
Parameters	NIP: allows to set the number of integration points (possible completions are 1, 8 (default), or 27).
Unknowns	Three dofs (u-displacement, v-displacement, w-displacement) are required in each node.
Approximation	Linear approximation of displacement and geometry.
Integration	Full integration of all strain components.
Features	Adaptivity support, Geometric nonlinearity support.
CS properties	-
Loads	-
Nlgeo	0,1,2.
Status	Reliable

Table 25: lspace element summary

2.8.2 LSpaceBB element

Implementation of 3d brick eight - node linear approximation element with selective integration of deviatoric and volumetric strain contributions (B-bar formulation) for incompressible problems. Features and description identical to conventional lspace element, see section 2.8.1.

2.8.3 QSpace element

Implementation of quadratic 3d 20-node finite element. Each node has 3 degrees of freedom. The element features are summarized in Table 26.

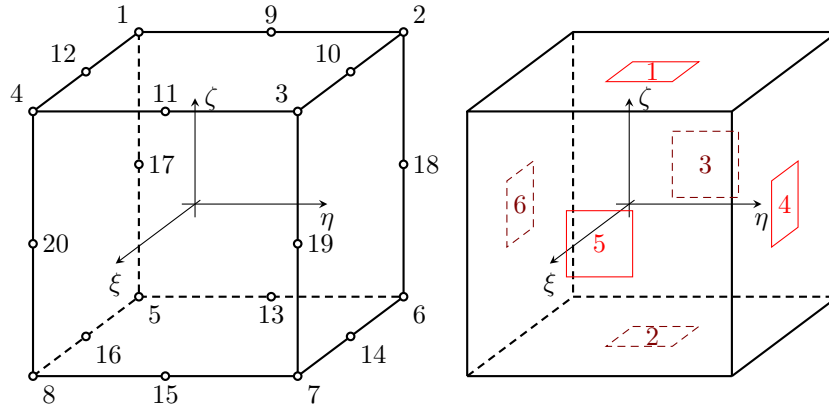


Figure 13: QSpace element.

Keyword	qspace
Description	Quadratic isoparametric brick element
Specific parameters	[NIP # _(in)]
Parameters	NIP: allows to set the number of integration points (possible completions are 1, 8 (default), or 27).
Unknowns	Three dofs (u-displacement, v-displacement, w-displacement) are required in each node.
Approximation	Quadratic approximation of displacement and geometry.
Integration	Full integration of all strain components.
Features	-
CS properties	-
Loads	-
Nlgeo	0,1,2.
Status	Reliable

Table 26: qspace element summary

2.8.4 LTRSpace element

Implementation of tetrahedra four-node finite element. Each node has 3 degrees of freedom. The element features are summarized in Table 27. Following node numbering convention is adopted (see also Fig. 14):

- Select a face that will contain the first three corners. The excluded corner will be the last one.
- Number these three corners in a counterclockwise sense when looking at the face from the excluded corner.

2.8.5 QTRSpace element

Implementation of tetrahedra ten-node finite element. Each node has 3 degrees of freedom. The element features are summarized in Table 28. Following node numbering convention is adopted (see also Fig. 15):

2.8.6 LWedge element

Implementation of wedge six-node finite element. Each node has 3 degrees of freedom. The element features are summarized in Table 29. Following node numbering convention is adopted (see also Fig. 16):

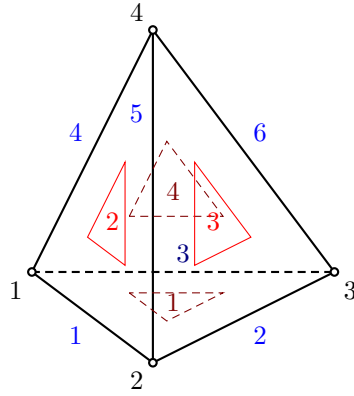


Figure 14: LTRSpace element. Definition and node numbering convention.

Keyword	LTRSpace
Description	Linear tetrahedra element
Specific parameters	-
Unknowns	Three dofs (u-displacement, v-displacement, w-displacement) are required in each node.
Approximation	Linear approximation of displacements and geometry using linear volume coordinates.
Integration	Full integration of all strain components using four point Gauss integration formula.
Features	Adaptivity support, Geometric nonlinearity support.
CS properties	-
Loads	Surface and Edge loadings supported.
Nlgeo	0,1,2.
Status	Reliable

Table 27: LTRSpace element summary

2.8.7 QWedge element

Implementation of wedge fifteen-node finite element. Each node has 3 degrees of freedom. The element features are summarized in Table 30. Following node numbering convention is adopted (see also Fig. 17):

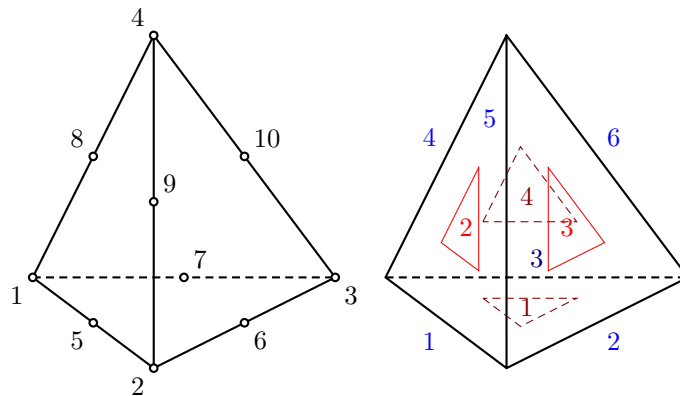


Figure 15: QTRSpace element. Definition and node numbering convention.

Keyword	QTRSpace
Description	3D tetrahedra element with quadratic interpolation
Specific parameters	[NIP # _(in)]
Parameters	NIP: allows to alter the default integration formula (possible completions are 1, 4 (default), 5, 11, 15, 24, and 45 point intergartion formulas).
Unknowns	Three dofs (u-displacement, v-displacement, w-displacement) are required in each node.
Approximation	Quadratic approximation of displacements and geometry using linear volume coordinates.
Integration	Full integration of all strain components using four point Gauss integration formula.
Features	-
CS properties	-
Loads	-
Nlgeo	0,1,2.
Status	Reliable

Table 28: QTRSpace element summary

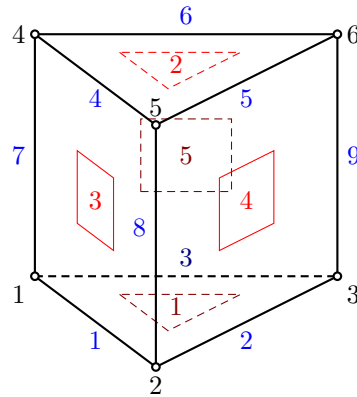


Figure 16: LWedge element. Node numbering convention in black, edge numbering in blue and face numbering in red.

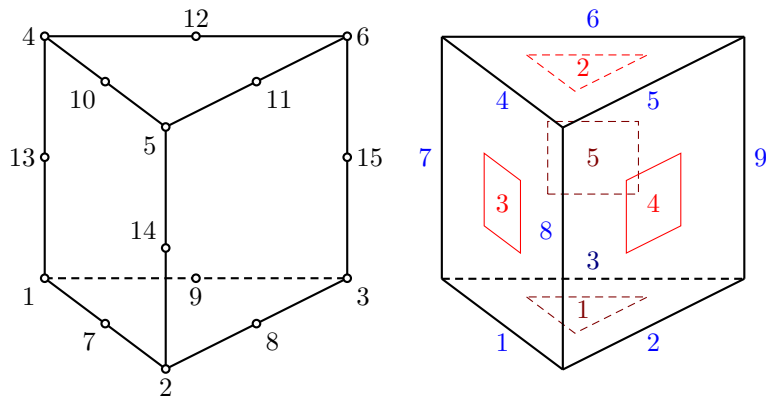


Figure 17: QWedge element. Node numbering convention in black, edge numbering in blue and face numbering in red.

Keyword	LWedge
Description	3D wedge six-node finite element with linear interpolation
Specific parameters	[NIP # _(in)]
Parameters	NIP: allows to alter the default integration formula (possible completions are 2 (default) and 9 point integration formulas).
Unknowns	Three dofs (u-displacement, v-displacement, w-displacement) are required in each node.
Approximation	Linear approximation of displacements and geometry.
Integration	Full integration of all strain components using four point Gauss integration formula.
Features	-
CS properties	-
Loads	-
Nlgeo	0,1,2.
Status	Reliable

Table 29: LWedge element summary

Keyword	QWedge
Description	3D wedge six-node finite element with quadratic interpolation
Specific parameters	[NIP # _(in)]
Parameters	NIP: allows to alter the default integration formula (possible completions are 2 (default) and 9 point integration formulas).
Unknowns	Three dofs (u-displacement, v-displacement, w-displacement) are required in each node.
Approximation	Quadratic approximation of displacements and geometry.
Integration	Full integration of all strain components using four point Gauss integration formula.
Features	-
CS properties	-
Loads	-
Nlgeo	0,1,2.
Status	Reliable

Table 30: QWedge element summary

2.9 Interface elements

Interface elements represent a contact between two adjacent nodes. Interface material specific to interface element needs to be assigned - consult the *matlibmanual* for interface support.

2.9.1 Interface1d element

Implementation of one dimensional (slip) interface element. This element can connect two separate nodes by specifying the one-dimensional slip law, that determines the force acting between these nodes depending on their relative displacement. This element can be applied in 1D, 2D, and 3D settings (default). The element features are summarized in Table 31.

Keyword	Interface1d
Description	One dimensional (slip) interface element
Specific parameters	[refnode # _(in)] [normal # _(ra)]
Parameters	refnode : determines the reference node, which is used to specify a reference direction (the direction vector is obtained by subtracting the coordinates of the first node from the reference node). normal : The reference direction can be directly specified by the optional parameter normal . Although both refnode and normal are optional, at least one of them must be specified.
Unknowns	One, two, or three DOFs (u-displacement, v-displacement, w-displacement) are required in each node, according to element mode (determined from domain type).
Approximation	-
Integration	-
Features	-
CS properties	-
Loads	-
Nlgeo	0
Status	Reliable
Note	Element requires material model with <code>_1dInterface</code> support.

Table 31: Interface1d element summary

2.9.2 Interface2dquad element

Implementation of a two dimensional interface element with quadratic approximation of displacement field. Can be used to glue together two elements with quadratic displacement approximation along the shared edge. Note, that the nodes along the interface are doubled, each couple with identical coordinates. Nodes on the negative side are numbered first, followed by nodes on the positive part. Requires material model with `_2dInterface` support. The element features are summarized in Table 32.

2.9.3 Interface3dtrlin element

Implementation of a three dimensional interface element with linear approximation of displacement field. Can be used to glue together two elements with linear displacement approximation along the shared triangular surface. Note, that the nodes along the interface are doubled, each couple with identical coordinates. Nodes on the negative surface are numbered first, followed by nodes on the positive part. The numbering of surface nodes on interface determines the positive normal (right hand rule). The surface in the direction of positive

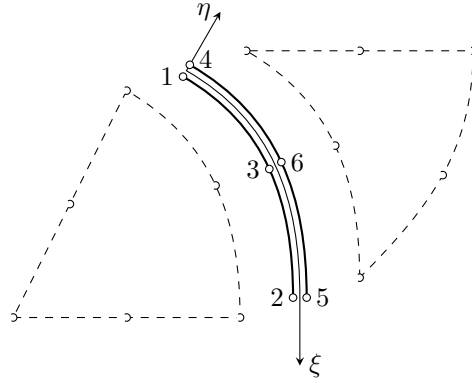


Figure 18: Interface2dquad element with quadratic interpolation. Definition and node numbering convention

Keyword	Interface2dquad
Description	2D interface element with quadratic approximation
Specific parameters	-
Unknowns	Two dofs (u-displacement, v-displacement) are required in each node.
Approximation	Quadratic approximation of displacements and geometry.
Integration	Full integration of all strain components using four point integration formula.
Features	-
CS properties	-
Loads	-
Nlgeo	0
Status	Reliable
Note	Element requires material model with .2dInterface support.

Table 32: Interface2dquad element summary

normal is the positive surface. Requires material model with .3dInterface support. The element features are summarized in Table 33.

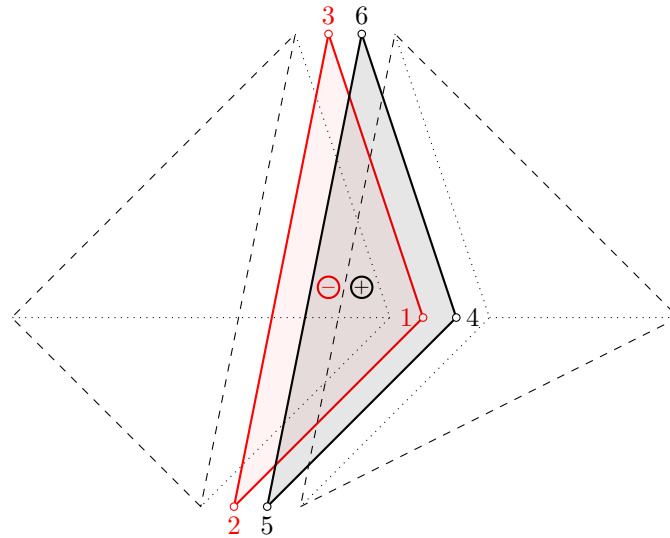


Figure 19: Interface3dtrlin element with linear interpolation. Definition and node numbering convention

Keyword	Interface3dtrlin
Description	3D interface element with linear approximation
Specific parameters	-
Unknowns	Three dofs (u-displacement, v-displacement, w-displacement) are required in each node.
Approximation	Linear approximation of displacements and geometry.
Integration	Full integration of all components using one point integration formula.
Features	-
CS properties	-
Loads	-
Nlgeo	0
Status	Reliable
Note	Element requires material model with .3dInterface support.

Table 33: Interface3dtrlin element summary

2.10 XFEM elements

XFEM elements allow simulations where the unknown fields are enriched through the partition of unity concept. Two elements are currently available for 2D XFEM simulations: *TrPlaneStress2dXFEM* (subclass of *TrPlaneStress2d*) and *PlaneStress2dXfem* (subclass of *PlaneStress2d*).

2.10.1 TrPlaneStress2dXFEM element

Keyword	TrPlaneStress2dXFEM
Description	Two dimensional 3-node triangular XFEM element
Specific parameters	[czmaterial # _(in)] [nipc # _(in)][useplanestrain # _(in)]
Parameters	czmaterial : Interface material for the cohesive zone. (If no material is specified, traction free crack surfaces are assumed.) nipc : Number of integration points used on each segment of the cohesive zone. useplanestrain : If plane strain or plane stress should be assumed. 0 implies plane stress and 1 implies plane strain.
Unknowns	Two continuous (standard) DOFs (u-displacement, v-displacement) and a variable number of enriched DOFs (can be continuous or discontinuous).
Approximation	-
Integration	Elements cut by an XFEM interface are divided into subtriangles.
Features	-
CS properties	-
Loads	-
Nlgeo	-
Status	-
Note	Test case: sm/xFemCrackVal.in

Table 34: TrPlaneStress2dXFEM element summary

2.10.2 PlaneStress2dXfem element

Keyword	PlaneStress2dXfem
Description	Two dimensional 4-node quad XFEM element
Specific parameters	[czmaterial # _(in)] [nipcz # _(in)][useplanestrain # _(in)]
Parameters	czmaterial : Interface material for the cohesive zone. (If no material is specified, traction free crack surfaces are assumed.)
Unknowns	nipcz : Number of integration points used on each segment of the cohesive zone.
Approximation	useplanestrain : If plane strain or plane stress should be assumed. 0 implies plane stress and 1 implies plane strain.
Integration	Two continuous (standard) DOFs (u-displacement, v-displacement) and a variable number of enriched DOFs (can be continuous or discontinuous).
Features	-
CS properties	-
Loads	-
Nlgeo	-
Status	-
Note	Test cases: sm/xFemCrackValBranch.in, sm/xfemCohesiveZone1.in, benchmark/xfem01.in

Table 35: PlaneStress2dXfem element summary

2.11 Iso Geometric Analysis based (IGA) elements

The following record describes the common part of IGA element record:

```
*IGAElement (num#)(in)
    mat #(in) crossSect #(in) nodes #(ia)
    knotvectoru #(ra) knotvectorv #(ra) knotvectorw #(ra)
    [knotmultiplicityu #(ia)] [knotmultiplicityv #(ia)]
    [knotmultiplicityw #(ia)]
    degree #(ia) nip #(ia)
    <[partitions #(ia)] <[remote #()]
```

The **knotvectoru**, **knotvectorv**, and **knotvectorw** parameters specify knot vectors in individual parametric directions, considering only distinct knots. Open knot vector is always assumed, so the multiplicity of the first and last knot should be equal to $p + 1$, where p is polynomial degree in corresponding direction (determined by **degree** parameter, see further).

The knot multiplicity can be set using optional parameters **knotmultiplicityu**, **knotmultiplicityv**, and **knotmultiplicityw**. By default, the open knot vector is assumed and multiplicity of internal knots is assumed to be equal to one. Note, that total number of knots in particular direction (including multiplicity) must be equal to number of control points in this direction increased by degree in this direction plus 1.

The degree of approximation for each parametric direction is determined from **degree** array, dimension of which is equal to number of spatial dimensions of the problem.

In case of elements with BSpline or Nurbs interpolation, the nodes forming the rectangular array of control points of the element are ordered in a such way, that u-index is changing most quickly, and w-index (or v-index in case of 2d problems) most slowly. In case of elements with T-spline interpolation, the nodes forming the T-mesh of the element are ordered arbitrarily.

The supported ***IGAElement** values are following:

Keyword: bsplineplanestresselement

Parameters: None.

Keyword: nurbsplanestresselement

Parameters: None.

Keyword: nurbs3delement

Parameters: None.

Keyword: tsplineplanestresselement

Parameters: localindexknotvectoru #_(in) localindexknotvectorv #_(in) localindexknotvectorw #_(in)

The parameters **localindexknotvectoru**, **localindexknotvectorv**, and **localindexknotvectorw** defined by the indices to global knot vectors (given by **knotvectoru**, **knotvectorv**, and **knotvectorw** parameters) specify the local knot vectors for each control point of T-mesh (node) in the same order as the nodes have been specified for the element. The local knot vector in a particular direction has $p + 2$ entries, where the p is the polynomial degree in that direction.

2.12 Special elements

2.12.1 LumpedMass element

This element, defined by a single node, allows to introduce additional concentrated mass and/or rotational inertias in a node. A different mass and rotary inertia may be assigned to each coordinate direction. At present, individual mass/inertia components can be specified for every degree of freedom of element node. Only displacement and rotational degrees of freedom are considered. The element features are summarized in Table 36.

Keyword	LumpedMass
Description	Lumped mass element
Specific parameters	components $\#_{(ra)}$
Parameters	components: allows to specify additional concentrated mass components ($\text{Force} \cdot \text{Time}^2 / \text{Length}$) and rotary inertias ($\text{Force} \cdot \text{Length} \cdot \text{Time}^2$) about the nodal coordinate axes. Only displacement and rotational degrees of freedom are considered. The individual DOFs are ordered according to following rule: first, displacement degrees of freedom in the direction of x,y, and z axes (if any), followed by rotational DOFs around x,y, and z axes (if any).
Unknowns	Only displacement and rotational DOFs are considered (D_u , D_v , D_w , R_u , R_v , and R_w).
Approximation	-
Integration	-
Features	-
CS properties	-
Loads	-
Status	Reliable

Table 36: LumpedMass element summary

2.12.2 Spring element

This element represent longitudinal or torsional spring element. It is defined by two nodes, orientation and a spring constant. The spring element has no mass associated, the mass can be added using LumpedMass element. The spring is linear and works the same way in tension or in compression. The element features are summarized in Table 37.

Keyword	Spring
Description	Spring element
Specific parameters	mode $\#_{(in)}$ k $\#_{(rn)}$ orientation $\#_{(ra)}$
Parameters	mode : defines the type of spring element (see Table 38). k : determines the spring constant, corresponding units are [Force/Length] for longitudinal spring and [Force*Length/Radian] for torsional spring. orientation : defines orientation vector of spring element (of size 3) - for longitudinal spring it defines the direction of spring, for torsional spring it defines the axis of rotation. the spring element nodes doesn't need to be coincident, but the spring orientation is always determined by orientation vector.
Note	

Table 37: Spring element summary

mode	description
0	1D spring element along x-axis, requires D_u DOF in each node, orientation vector is {1,0,0}
1	2D spring element in xy plane, requires D_u and D_v DOFs in each node (orientation vector should be in xy plane)
2	2D torsional spring element in xz plane, requires R_v DOFs in each node
3	3D spring element in space, requires D_u, D_v, and D_w DOFs in each node
4	3D torsional spring in space, requires R_u, R_v, and R_w DOFs in each node

Table 38: Supported spring element modes

nlgeo	strain tensor
0 (default)	Small-strain tensor
1	Green-Lagrange strain tensor
2	Deformation gradient

Table 39: Nonlinear geometry modes

2.13 Geometric nonlinear analysis

To take geometric nonlinearity into account on an element, keyword **nlgeo** has to be specified. The **nlgeo** parameter defines which type of the strain tensor is computed (see Table 39).

3 Elements for Transport problems (TM Module)

3.1 2D Elements

3.1.1 Quad1ht element

Represents isoparametric four-node quadrilateral finite element for heat transfer problems. Each node has 1 degree of freedom. Problem should be defined in x,y plane. The cross section thickness property is requested form cross section model. The nodes should be numbered anti-clockwise (positive rotation around z-axis). The element features are summarized in Table 40.

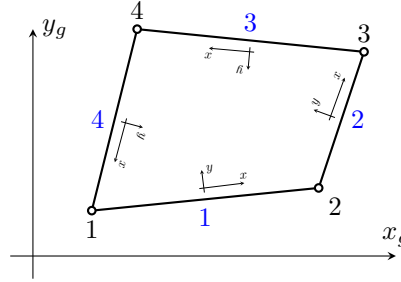


Figure 20: Quad1ht element. Node numbering, Side numbering and definition of local edge c.s.(a).

Keyword	Quad1ht
Description	Isoparametric four-node quadrilateral linear interpolation element for heat transfer problems
Specific parameters	[NIP # _(in)]
Parameters	NIP: allows to change the default number of integration point used.
Unknowns	Single dof (T _f - temperature) is required in each node.
Approximation	Linear approximation of temperature.
Integration	Integration using gauss integration formula in 4 (the default), 9, or 16 integration points. The default number of integration point used can be overloaded using NIP parameter.
Loads	Body loads are supported. Boundary loads are supported and computed using numerical integration. The side numbering is following. Each i-th element side begins in i-th element node and ends on next element node (i+1-th node or 1-st node, in the case of side number 4). The local positive edge x-axis coincides with side direction, the positive local edge y-axis is rotated 90 degrees anti-clockwise (see fig. (20)).
Features	-
CS properties	-
Status	

Table 40: Quad1ht element summary

3.1.2 Quad1hmt element

Represents isoparametric four-node quadrilateral finite element for heat and mass (one constituent) transfer problems. Two dofs (T_f - temperature and C₁ - concentration) are required in each node. Linear approximation of temperature and mass concentration. Other features are similar to Quad1 element, see section 3.1.1.

3.1.3 Tr1ht element

Implements the linear triangular finite element for heat transfer problems. Each node has 1 degree of freedom. The cross section thickness property is requested from cross section model. The node numbering is anti-clockwise. The element features are summarized in Table 41.

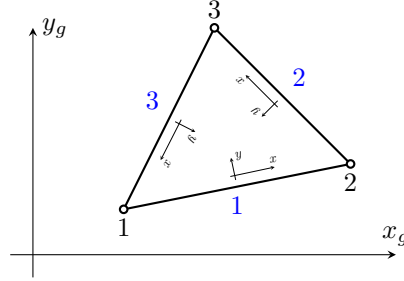


Figure 21: Tr1ht element - node and side numbering.

Keyword	Tr1ht
Description	triangular finite element with linear approximation for heat transfer problems
Specific parameters	-
Unknowns	Single dof (T.f - temperature) is required in each node.
Approximation	Linear approximation of temperature.
Integration	Integration using one point gauss integration formula.
Loads	Body loads are supported. Boundary loads are supported and are computed using numerical integration. The side numbering is following. Each i-th element side begins in i-th element node and ends on next element node (i+1-th node or 1-st node, in the case of side number 3). The local positive edge x-axis coincides with side direction, the positive local edge y-axis is rotated 90 degrees anti-clockwise (see fig. (21)).
Features	-
CS properties	-
Status	

Table 41: Tr1ht element summary

3.2 Axisymmetric Elements

3.2.1 Quadaxisym1ht element

Isoparametric four-node quadrilateral finite element for axisymmetric heat transfer problems. The element description is similar to Quad1 element, see section 3.1.1.

3.2.2 Traxisym1ht element

Linear triangular finite element for axisymmetric heat transfer problems. The element description is similar to Tr1ht element, see section 3.1.3.

3.3 3D Elements

3.3.1 Tetrah1ht - tetrahedral 3D element

Represents isoparametric four-node tetrahedral element. Each node has 1 degree of freedom. The same numbering convention is adopted as in mechanics, see Fig. 14. The element features are summarized in Table 42.

Keyword	Tetrah1ht
Description	Isoparametric, four-node tetrahedral element with linear approximation for heat transfer problems
Specific parameters	[NIP # _(in)]
Parameters	NIP: allows to change the default number of integration point used.
Unknowns	Single dof (T.f - temperature) is required in each node.
Approximation	Linear approximation of temperature.
Integration	Integration using gauss integration formula in 1 (the default), or 4 integration points. The default number of integration point used can be overloaded using NIP parameter.
Loads	Body loads are supported. Boundary loads are supported and computed using numerical integration. The side and surface numbering is shown in Fig. 14.
Features	-
CS properties	-
Status	

Table 42: Tetrah1ht element summary

3.3.2 Brick1ht - hexahedral 3D element

Represents isoparametric eight-node brick/hexahedron finite element for heat transfer problems. Each node has 1 degree of freedom. The element features are summarized in Table 43.

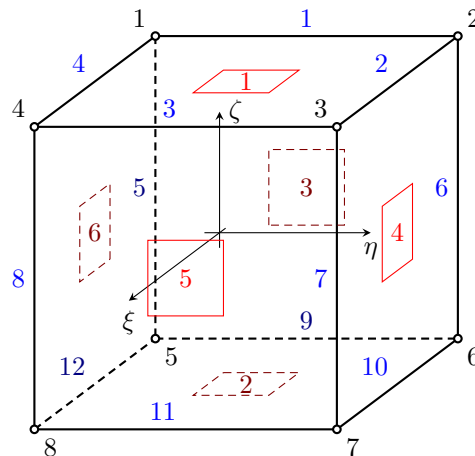


Figure 22: Brick1ht element. Node numbers are in black, side numbers are in blue, and surface numbers are in red.

Keyword	Brick1ht
Description	Isoparametric, hexahedral 3D element with linear approximation for heat transfer problems
Specific parameters	[NIP # _(in)]
Parameters	NIP: allows to change the default number of integration point used.
Unknowns	Single dof (T_f - temperature) is required in each node.
Approximation	Linear approximation of temperature.
Integration	Integration using gauss integration formula in 8 (the default), or 27 integration points. The default number of integration point used can be overloaded using NIP parameter.
Loads	Body loads are supported. Boundary loads are supported and computed using numerical integration. The side and surface numbering is shown in fig. (22)).
Features	-
CS properties	-
Status	

Table 43: Brick1ht element summary

3.3.3 Brick1hmt - hexahedral 3D element

Represents isoparametric eight-node quadrilateral finite element for heat and mass (one constituent) transfer problems. Two dofs (T_f - temperature and C_1 - concentration) are required in each node. Linear approximation of temperature and mass concentration. Other features are similar to Brick1 element, see section 3.3.2.

3.3.4 QBrick1ht - quadratic hexahedral 3D element

Implementation of quadratic 3d 20-node finite element. Each node has 1 degree of freedom. See section 2.8.3 for node numbering order and order of faces. The element features are summarized in Table 44.

Keyword	QBrick1ht
Description	Isoparametric, hexahedral 3D element with quadratic approximation for heat transfer problems
Specific parameters	[NIP # _(in)]
Parameters	NIP: allows to change the default number of integration point used, possible values are 8, 27 (default) and 64.
Unknowns	Single dof (T_f - temperature) is required in each node.
Approximation	Quadratic approximation of temperature and geometry..
Integration	Integration using gauss integration formula in 8, 27 (default), or 64 integration points. The default number of integration point used can be overloaded using NIP parameter.
Loads	-
Features	-
CS properties	-
Status	

Table 44: QBrick1ht element summary

3.3.5 QBrick1hmt - quadratic hexahedral 3D element

The same element as QBrick1ht for heat and mass (one constituent) transfer problems. Two dofs (T_f - temperature and C_1 - concentration) are required in each node. Linear approximation of temperature and

mass concentration. Other features are similar to QBrick1ht element, see section 3.3.4.

4 Elements for Fluid Dynamics problems (FM Module)

4.1 Stokes' Flow Elements

Stokes' flow elements neglect acceleration, and thus requires no additional stabilization.

4.1.1 Tr21Stokes element

Standard 6 node triangular element for stokes flow, with quadratic geometry, velocity and linear pressure. Both compressible and incompressible material behavior is supported (and also the seamless transition between the two). The element features are summarized in Table 45.

Keyword	Tr121Stokes
Description	Standard 6 node triangular element for stokes flow, with quadratic geometry, velocity and linear pressure
Specific parameters	-
Unknowns	Unknown pressure in nodes 1–3 with unknown velocity (V_u and V_v) in all 6 nodes.
Approximation	Quadratic approximation of geometry and velocity, linear pressure approximation.
Integration	
Features	-
Status	Reliable

Table 45: Tr121Stokes element summary

4.1.2 Tet21Stokes element

Standard 10 node tetrahedral element for stokes flow, with quadratic geometry, velocity and linear pressure. The element features are summarized in Table 46.

Keyword	Tet21Stokes
Description	Standard 10 node tetrahedral element for stokes flow, with quadratic geometry, velocity and linear pressure
Specific parameters	-
Unknowns	Unknown pressure (P_f) in nodes 1–4, and unknown velocity (V_u , V_v , V_w) in all nodes.
Approximation	Quadratic approximation of geometry and velocity, linear pressure approximation.
Integration	
Features	-
Status	Untested

Table 46: Tet21Stokes element summary

4.1.3 Hexa21Stokes element

Standard 27 node hexahedral element for stokes flow, with quadratic geometry, velocity and linear pressure. The element features are summarized in Table 47.

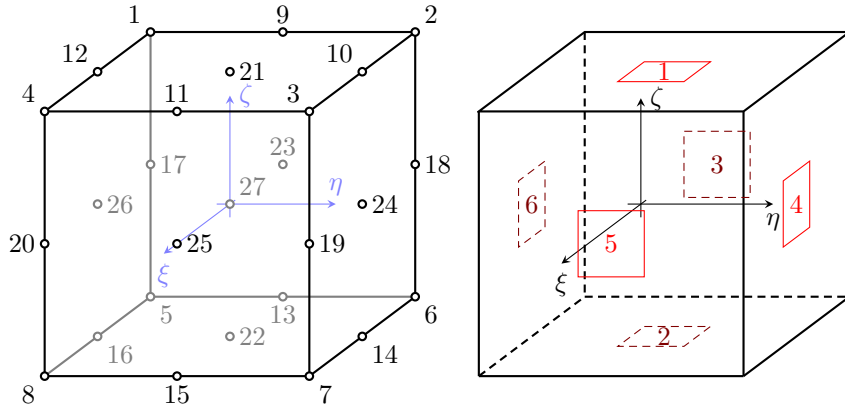


Figure 23: Hexa21Stokes element. Node numbering and face numbering.

Keyword	Hexa21Stokes
Description	Standard 10 node tetrahedral element for stokes flow, with quadratic geometry, velocity and linear pressure
Specific parameters	-
Unknowns	Unknown pressure (P_f) in nodes 1–8, and unknown velocity (V_u, V_v, V_w) in all nodes.
Approximation	Quadratic approximation of geometry and velocity, linear pressure approximation.
Integration	-
Features	-
Status	Untested

Table 47: Hexa21Stokes element summary

4.1.4 Tr1BubbleStokes element

So called “Mini” element in 2D. A 3 node triangular element for stokes flow, with linear geometry and pressure. Velocity is enriched by a bubble function. Should not be used with materials that have memory (which is uncommon for flow problems). The element features are summarized in Table 48.

Keyword	Tr1BubbleStokes
Description	So called “Mini” 2D element
Specific parameters	-
Unknowns	Unknown pressure (P_f) in all nodes, and unknown velocity (V_u, V_v) in all nodes and one internal dof manager.
Approximation	Linear geometry and pressure. Velocity is enriched by a bubble function.
Integration	-
Features	-
Status	Untested

Table 48: Tr1BubbleStokes element summary

4.1.5 Tet1BubbleStokes element

So called “Mini” element in 3D. A 4 node tetrahedral element for stokes flow, with linear geometry and pressure. Velocity is enriched by a bubble function. Should not be used with materials that have memory (which is

uncommon for flow problems). The element features are summarized in Table 49.

Keyword	Tet1BubbleStokes
Description	So called “Mini” 3D element
Specific parameters	-
Unknowns	Unknown pressure (P_f) in all nodes, and unknown velocity (V_u, V_v) in all nodes and one internal dof manager.
Approximation	Linear geometry and pressure. Velocity is enriched by a bubble function.
Integration	
Features	-
Status	Untested

Table 49: Tet1BubbleStokes element summary

4.2 2D CBS Elements

4.2.1 Tr1CBS element

Represents the linear triangular finite element for transient incompressible flow analysis using cbs algorithm with equal order approximation of velocity and pressure fields. Each node has 3 degrees of freedoms (two components of velocity and pressure). The node numbering is anti-clockwise. The element features are summarized in Table 50.

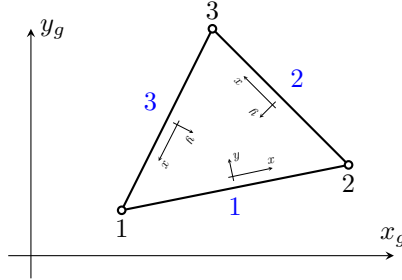


Figure 24: Tr1CBS element. Node numbering, Side numbering and definition of local edge c.s.(a).

Keyword	Tr1CBS
Description	linear triangular finite element for transient incompressible flow analysis using cbs algorithm
Specific parameters	[bsides $\#_{(ia)}$] [bcodes $\#_{(ia)}$]
Parameters	Since the problem formulation requires to evaluate some boundary terms, the element boundary edges should be specified as well as the types of boundary conditions applied at these boundary edges. The boundary edges (their numbers) are specified using bsides array. The type of boundary condition(s) applied to corresponding boundary side is determined by bcodes array. The available/supported boundary codes are following: 1 for prescribed traction, 2 for prescribed normal velocity, 4 for prescribed tangential velocity, and 8 for prescribed pressure. If the element side is subjected to a combination of these fundamental types boundary conditions, the corresponding code is obtained by summing up the corresponding codes.
Unknowns	Two velocity components (V_u and V_v) and pressure (P_f) are required in each node.
Approximation	Equal order approximation of velocity and pressure fields.
Integration	exact
Features	Constant boundary tractions are supported ¹ . Body loads representing the self-weight load are supported.
Status	Untested

Table 50: Tr1CBS element summary

4.3 2D SUPG/PSPG Elements

4.3.1 Tr1SUPG element

Represents the linear triangular finite element for transient incompressible flow analysis using SUPG/PSPG stabilization with equal order approximation of velocity and pressure fields. Each node has 3 degrees of freedoms (two components of velocity and pressure). The node numbering is anti-clockwise. The element features are summarized in Table 51.

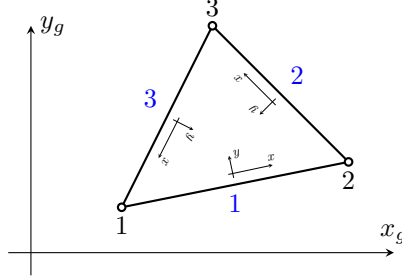


Figure 25: Tr1SUPG element. Node numbering, Side numbering and definition of local edge c.s.(a).

Keyword	Tr1SUPG
Description	linear triangular finite element for transient incompressible flow analysis using SUPG/PSPG algorithm
Specific parameters	[vo f # _(rn)] [p vo f # _(rn)]
Unknowns	Two velocity components (V_u and V_v) and pressure (P_f) are required in each node.
Approximation	Linear approximation of velocity and pressure fields.
Integration	exact
Loads	Constant boundary tractions are supported. Body loads representing the self-weight load are supported.
Multi-fluid analysis	The element has support for solving problems with two immiscible fluids in a fixed spatial domain. In the present implementation, a VOF and LevelSet tracking algorithms are used to track the position of interface. In case of VOF tracking, an initial VOF fraction (volume fraction of reference fluid) can be specified using vo f (default is zero). Element can also be marked as allways filled with reference fluid (some form of source) using parameter p vo f which specifies the permanent VOF value. In case of LevelSet tracking, the initial levelset is specified using reference polygon (see corresponding levelset record in oofem input manual). The material model should be of type Keyword: twofluidmat , that supports modelling of two immiscible fluids.
Status	Reliable

Table 51: Tr1SUPG element summary

4.3.2 Tr21SUPG element

Implementation of P2P1 Taylor Hood element for transient incompressible flow analysis using SUPG and LSIC stabilization. It consists of globally continuous, piecewise quadratic functions for approximation in velocity space and globally continuous, piecewise linear functions for approximation in pressure space. LBB condition is

satisfied. There are 3 degrees of freedom in vertices (two components of velocity and pressure), and 2 degrees of freedom in edge nodes (two components of velocity only). The node numbering is anti-clockwise, vertices are numbered first. The element features are summarized in Table 52.

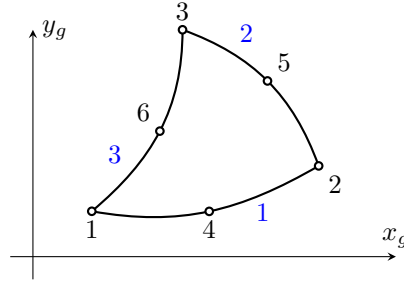


Figure 26: Tr21SUPG element - node and side numbering.

Keyword	Tr21SUPG
Description	P2P1 Taylor Hood element
Specific parameters	-
Unknowns	Two velocity components (V_u and V_v) and pressure (P_f) in vertices and two velocity components (V_u and V_v) in edge nodes are required.
Approximation	Quadratic approximation of velocity and linear approximation of pressure fields.
Integration	Integration is exact, each submatrix of element stiffness matrix is evaluated in proper number of Gauss points. Submatrices connected with velocity are evaluated in 7 or 13 points, mixed velocity-pressure submatrices in 3 or 7 points, submatrices connected with pressure in 3 points.
Loads	Constant boundary tractions are supported. Body loads representing the self-weight load are supported.
Multi-fluid analysis	The element has no support for solving problems with two immiscible fluids in a fixed spatial domain.
Status	Reliable

Table 52: Tr21SUPG element summary

4.3.3 Tr1SUPGAxi element

Represents the linear triangular finite element for transient incompressible flow analysis using SUPG/PSPG stabilization with equal order approximation of velocity and pressure fields in 2d-axisymmetric setting. Each node has 3 degrees of freedoms (two components of velocity and pressure). The y-axis is axis of rotational symmetry. The node numbering is anti-clockwise. The element features are summarized in Table 53.

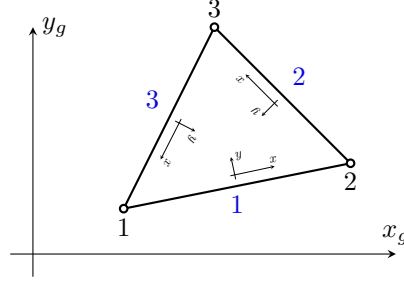


Figure 27: Tr1SUPGAxi element. Node numbering, Side numbering and definition of local edge c.s.(a).

Keyword	Tr1SUPGAxi
Description	linear equal order approximation axisymmetric element
Specific parameters	[vof # _(rn)][pvo # _(rn)]
Unknowns	Two velocity components (V_u and V_v) and pressure (P_f) are required in each node.
Approximation	Linear approximation of velocity and pressure fields.
Integration	Gauss integration in seven point employed.
Loads	Constant boundary tractions are supported. Body loads representing the self-weight load are supported.
Multi-fluid analysis	The element has support for solving problems with two immiscible fluids in a fixed spatial domain. In the present implementation, a VOF tracking algorithm is used to track the position of interface. An initial VOF fraction (volume fraction of reference fluid) can be specified using vof (default is zero). Element can also be marked as always filled with reference fluid (some form of source) using parameter pvo which specifies the permanent VOF value. In this case, the material model should be of type Keyword: twofluidmat , that supports modelling of two immiscible fluids.
Status	

Table 53: Tr1SUPGAxi element summary

4.4 3D SUPG/PSPG Elements

4.4.1 Tet1.3D_SUPG element

Represents 3D linear pyramid element for transient incompressible flow analysis using SUPG/PSPG stabilization with equal order approximation of velocity and pressure fields. Each node has 3 degrees of freedoms (two components of velocity and pressure). The element features are summarized in Table 54.

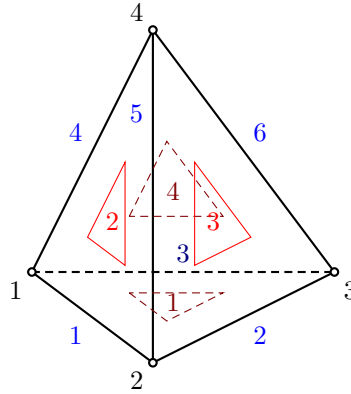


Figure 28: Tet1.3D_SUPG element.

Keyword	TET1SUPG
Description	linear equal order approximation axisymmetric element
Specific parameters	[vof # _(rn)][pvof # _(rn)]
Unknowns	Three velocity components (V_u, V_v, and V_w) and pressure (P_f) are required in each node.
Approximation	Linear approximation of velocity and pressure fields.
Integration	exact
Loads	Constant boundary tractions are supported. Body loads representing the self-weight load are supported.
Multi-fluid analysis	The element has support for solving problems with two immiscible fluids in a fixed spatial domain. In the present implementation, a LevelSet tracking algorithm is used to track the position of interface. The material model should be of type Keyword: twofluidmat , that supports modelling of two immiscible fluids.
Status	

Table 54: TET1SUPG element summary