OOFEM.ORG

Element Library Manual

Bořek Patzák

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

December 29, 2011

Contents

List of Figures

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

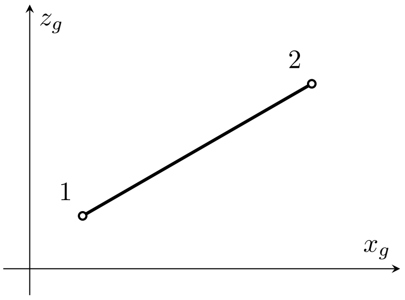

Figure 1: Truss2d element in (x,z) plane.

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.
Keyword: `truss1d`
Parameters: none.
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

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.
Keyword: `truss2d`
Parameters: [`cs`$_{\rm (in)}$ #]

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. The supported values of *cs* are following: 0 for (x,z) plane (default), 1 for (x,y) plane, and 3 for (y,z) plane.
Approximation: Linear approximation of displacements and geometry.

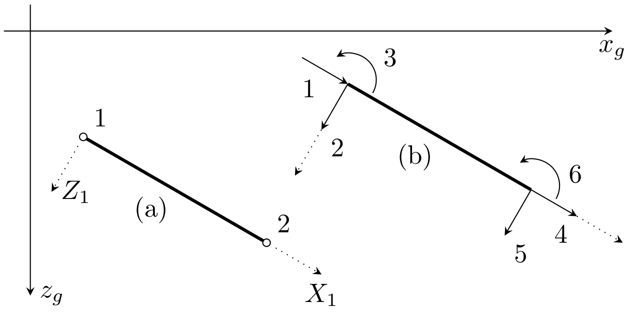

Figure 2: Beam2d element. Definition of local c.s.(a) and definition of local end forces and local element dofs (b).

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

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.
Keyword: `truss3d`
Parameters: none.

Unknowns: Three displacement DOFs (in x, y, and z directions) are required in each node.
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.
Status: Reliable

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.
Keyword: `beam2d`
Parameters: [`dofstocondense`$_{\text{(ia)}}$ #]
The *dofstocondense* parameter 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 (*sheararealz* 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.

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 $\boldsymbol{a_1} = (x_j - x_i, y_j - y_i, z_j - z_i)$,
3. local y-axis direction vector lies in plane defined by local x-axis direction vector $(\boldsymbol{a_1})$ 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 $(\boldsymbol{a_1})$ 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.

Keyword: `beam3d`
Parameters: `refnode`$_{\text{(in)}}$ # [`dofstocondense`$_{\text{(ia)}}$ #]
The *refnode* parameter determines the reference node. It determines the local coordinate system of beam element. The *dofstocondense* parameter 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

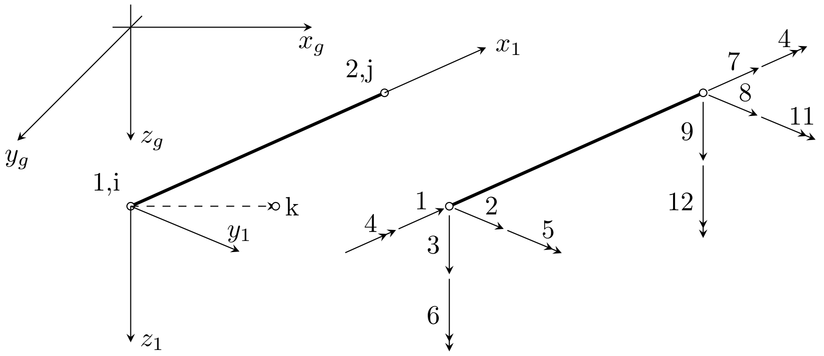

Figure 3: Beam3d element. Definition of local c.s., local end forces and local element dofs numbering.

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 (*beamshearcoeff* parameter) or equivalent shear areas (*sheareareay* and *sheareareaz* 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: Stable, various loadings require further testing.

2.3 Plane Stress Elements

2.3.1 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).
Keyword: `planestress2d`
Parameters: [`NIP`$_{\mathrm{(in)}}$ #]
Unknowns: Two dofs (u-displacement, v-displacement) are required in each node.
Approximation: Linear approximation of displacement 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 point used can be overloaded using *NIP* parameter. Reduced integration for shear terms is employed. Shear terms are always integrated

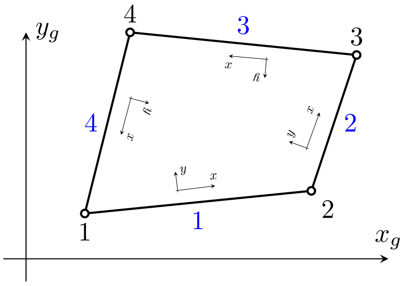

Figure 4: PlaneStress2d element. Node numbering, Side numbering and definition of local edge c.s.(a).

using 1 point integration rule.
Features: Nonlocal constitutive support, Geometric nonlinearity support.
CS properties: Thickness.
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. (4)).
Status: Reliable.

2.3.2 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. (5).

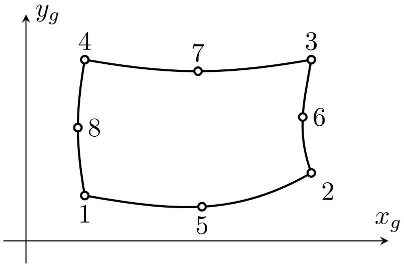

Figure 5: QPlaneStress element - node numbering.

Keyword: `qplanestress2d`
Parameters: [`NIP`(in) #]
Unknowns: Two dofs (u-displacement, v-displacement) are required in each node.
Approximation: Quadratic approximation of displacement and geometry.
Integration: Full 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.

Features: Adaptivity support.
CS properties: Thickness.
Loads: Body loads are supported. Boundary loads are not supported in current implementation.
Status: Stable.

2.3.3 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

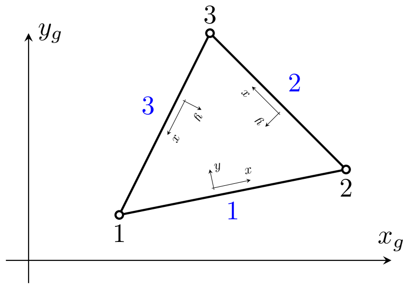

Figure 6: TrPlaneStress element - node and side numbering.

Keyword: `trplanestress2d`
Parameters: none.
Unknowns: Two dofs (u-displacement, v-displacement) are required in each node.
Approximation: Linear approximation of displacement 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: Thickness.
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. (6)).
Status: Reliable.

2.3.4 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. (7).
Keyword: `qtrplstr`
Parameters: [`NIP`$_{\mathrm{(in)}}$ #]
Unknowns: Two dofs (u-displacement, v-displacement) are required in each node.

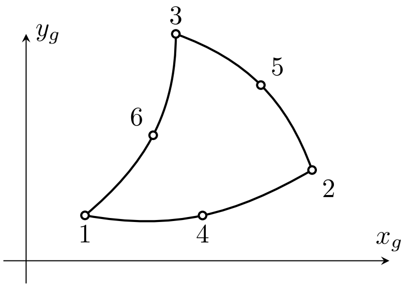

Figure 7: QTrPlStr element - node and side numbering.

Approximation: Quadratic approximation of displacement 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: Thickness.
Loads:
Status:

2.3.5 TrPlaneStrRot

Implementation of triangular three-node plane-stress finite element with independent rotation field. Each node has 3 degrees of freedom.
Keyword: `trplanestrrot`
Parameters: [`NIP`$_{\text{(in)}}$ #] [`NIPRot`$_{\text{(in)}}$ #]
Unknowns: Three dofs (u-displacement, v-displacement, z-rotation) are required in each node.
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).
CS properties: Thickness.
Features:
Status:

2.4 Plane Strain Elements

2.4.1 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).
Keyword: `quad1planestrain`
Parameters: [`NIP`$_{\text{(in)}}$ #]
Unknowns: Two dofs (u-displacement, v-displacement) are required in each node.
Approximation: Linear approximation of displacement and geometry.
Integration: Integration of membrane strain terms using gauss integration formula in 4 (the default), 9, or 16 integration points. The default number

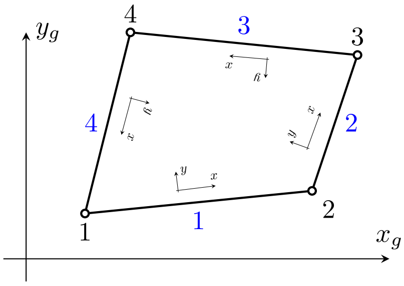

Figure 8: Quad1PlaneStrain element. Node numbering, Side numbering and definition of local edge c.s.(a).

of integration point 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: Thickness.
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. (8)).
Status: Reliable.

2.4.2 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

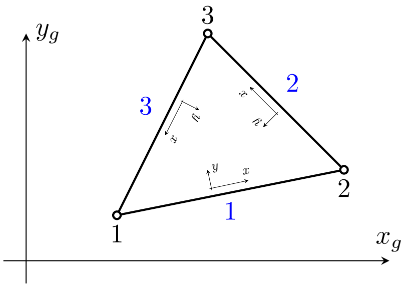

Figure 9: TrplaneStrain element - node and side numbering.

Keyword: `trplanestrain`
Parameters: none.
Unknowns: Two dofs (u-displacement, v-displacement) are required in each node.
Approximation: Linear approximation of displacement 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: Thickness.
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. (9)).
Status: Reliable.

2.5 Plate&Shell Elements

2.5.1 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).
Keyword: `cctplate`
Parameters: none.
Unknowns: Three dofs (w-displacement, u and v - rotation) are required in each node.
Integration: Integration of all terms using one point formula.
Features: Layered cross section support.
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.
Status: Reliable.

2.5.2 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).
Keyword: `cctplate3d`
Parameters: none.
Unknowns: Six dofs (u,v,w-displacements and u,v,w rotations) are required in each node.
Integration: Integration of all terms using one point formula.

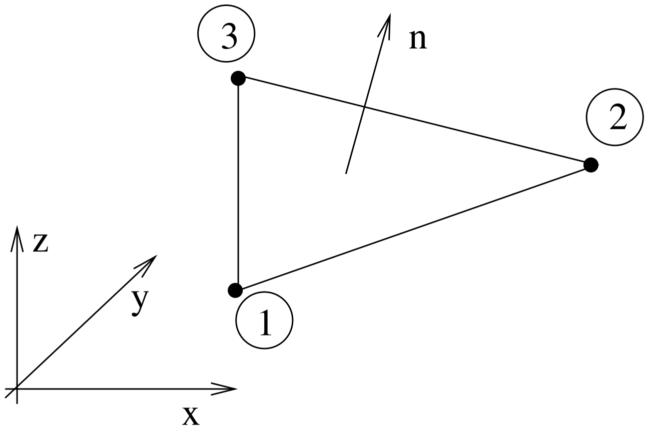

Figure 10: Geometry of tr_shell01 element.

Features: Layered cross section support.
Loads: Body loads are supported. Boundary loads are not supported. **Output**: On output, the generalized strains are printed in a vector with 12 components, with the same meaning as explained in section 2.5.1 for CCT element.
Status: Reliable.

2.5.3 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.
Keyword: `rershell`
Parameters: none.
Integration: Integration of all terms using one point formula.
Features: Layered cross section support.
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 same meaning as explained in section 2.5.1 for CCT element.
Status: Reliable.

2.5.4 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.
Keyword: `tr_shell01`
Parameters: none.
Integration: Integration of all terms using one point formula.
Features: Layered cross section support.
Loads: Body loads are supported. Boundary loads are not supported.
Output: On output, the generalized strains are printed in a vector with 12 components, with the same meaning as explained in section 2.5.1 for CCT element.

Status: Reliable.

2.6 Axisymmetric Elements

2.6.1 Axisymm3d element

Implementation of triangular three-node finite element for axisymmetric continuum. Each node has 2 degrees of freedom.
Keyword: `axisymm3d`
Parameters: [`NIP`$_{\text{(in)}}$ #] [`NIPfish`$_{\text{(in)}}$ #]
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 (default), 4 and 7 point integration rule). The remaining strain components (ε_ϕ and $\gamma_r z$) can be integrated using 1 (default), 4 and 7 integration point formulae (*NIPfish* parameter).
Features: None.
Loads: Boundary and body loads are supported.
Status:

2.6.2 Q4axisymm element

Implementation of quadratic isoparametric eight-node quadrilateral - finite element for axisimmetric 3d continuum. Each node has 2 degrees of freedom.
Keyword: `q4axisymm`
Parameters: [`NIP`$_{\text{(in)}}$ #] [`NIPfish`$_{\text{(in)}}$ #]
Unknowns: Two dofs (u-displacement, v-displacement) are required in each node.
Approximation: Quadratic approximation of displacements and geometry.
Integration: The integration of ε_x and ε_y strains can be altered using *NIP* parameter (possible completions are 1 (default), 4, 9 and 16 point integration rule). The remaining strain components (ε_ϕ and $\gamma_r z$) can be integrated using 1 (default), 4, 9 and 16 integration point formulae (*NIPfish* parameter).
Features: None.
Loads: No boundary and body loads are supported.
Status:

2.6.3 L4axisymm element

Implementation of isoparametric four-node quadrilateral axisymmetric finite element. Each node has 2 degrees of freedom.
Keyword: `l4axisymm`
Parameters: [`NIP`$_{\text{(in)}}$ #]
Unknowns: Two dofs (u-displacement, v-displacement) are required in each node.
Approximation: Linear approximation of displacements and geometry.
Integration: The integration of ε_x and ε_y strains can be altered using *NIP* parameter (possible completions are 1 (default), 4, 9 and 16 point integration rule). The remaining strain components (ε_ϕ and $\gamma_r z$) are integrated using one point integration formula.

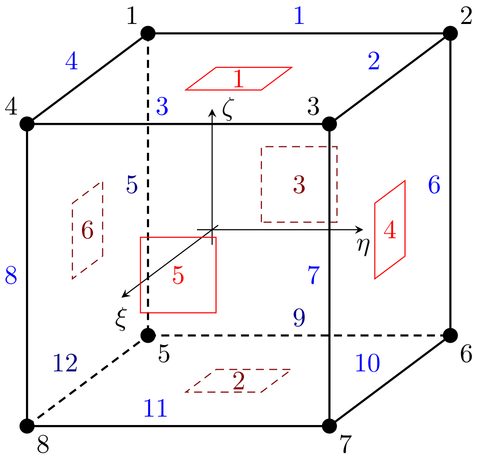

Figure 11: lspace element (Node numbers in black, side numbers in blue, and surface numbers in red).

Features: None.
Loads: No boundary and body loads are supported.
Status:

2.7 3d Continuum Elements

2.7.1 LSpace element

Implementation of Linear 3d eight - node finite element. Each node has 3 degrees of freedom.
Keyword: `lspace`
Parameters: [`NIP`$_{\text{(in)}}$ #]
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. *NIP* parameter (possible completions are 8 (default) and 27) allows to change the integration formula.
Features: Adaptivity support, Geometric nonlinearity support.
Loads:
Status:

2.7.2 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 2.7.1.

2.7.3 QSpace element

Implementation of quadratic 3d 20-node finite element. Each node has 3 degrees of freedom.

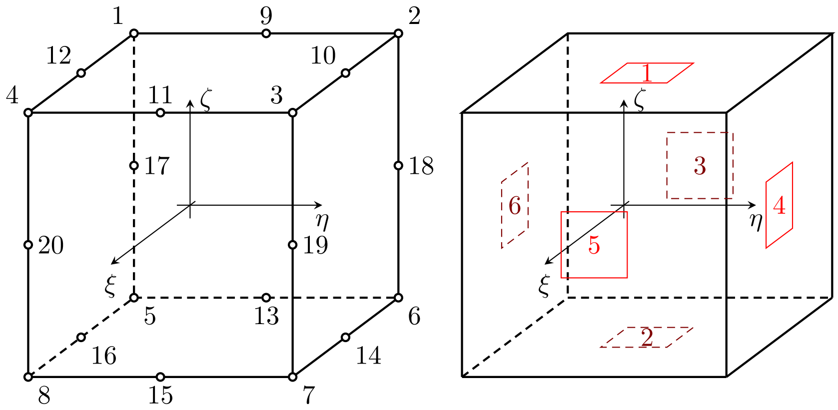

Figure 12: qspace element.

Keyword: `qspace`
Parameters: [`NIP`$_{\text{(in)}}$ #]
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. *NIP* parameter (possible completions are 8 (default) and 27) allows to change the integration formula.
Features: None.
Loads:
Status:

2.7.4 LTRSpace element

Implementation of tetrahedra four-node finite element. Each node has 3 degrees of freedom. Following node numbering convention is adopted (see also Fig. 13):

- 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.

Keyword: `LTRSpace`
Parameters: None.
Unknowns: Three dofs (displacement in x,y, and z axis directions) 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.
Loads: Surface and Edge loadings supported.
Status:

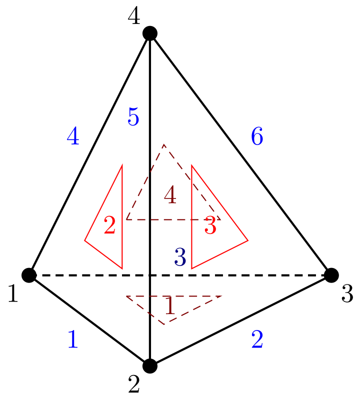

Figure 13: Linear terahedra element. Definition and node numbering convention

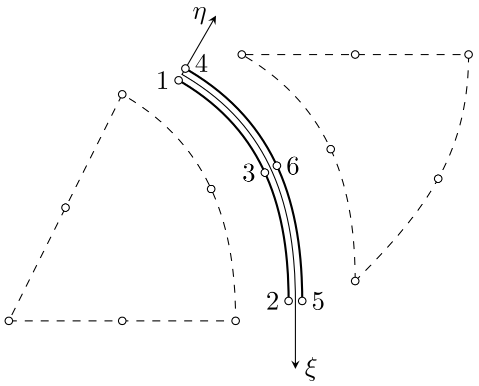

Figure 14: 2D interface element with quadratic interpolation. Definition and node numbering convention

2.8 Interface elements

2.8.1 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.
Keyword: `Interface2dquad`
Parameters: None.
Unknowns: Two 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 integration formula. **Features**: None.
Loads:

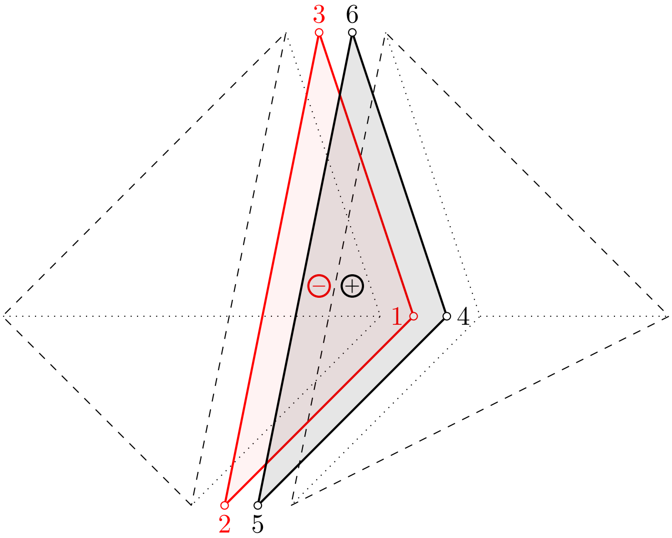

Figure 15: 3D interface element with linear interpolation. Definition and node numbering convention

Status:

2.8.2 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 normal is the positive surface. Requires material model with _3dInterface support.
Keyword: `Interface3dtrlin`
Parameters: None.
Unknowns: Two 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**: None.
Loads:
Status:

2.8.3 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).

Keyword: `Interface1d`
Parameters: [`refnode`$_{(in)}$ #] [`normal`$_{(ra)}$ #]
The optional parameter *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). 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. The magnitude of slip is then obtained as relative displacement vector of two element nodes projected to reference direction. As a consequence, this will lead to linearized geometrical equations, where slip is always related to the original (undeformed) configuration. Element requires material model with _1dInterface support. **Unknowns**: One, two, or three DOFs (u-displacement, v-displacement, w-displacement) are required in each node, according to element mode.
Approximation: None. **Integration**: None. **Features**: None.
Loads:
Status:

2.9 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 coresponding 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`$_{\text{(in)}}$ # `localindexknotvectorv`$_{\text{(in)}}$ # `localindexknotvectorw`$_{\text{(in)}}$ #
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.

2.10 Special elements

2.10.1 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 od freedom are considered.
Keyword: `LumpedMass`
Parameters: `components`$_{\text{(ra)}}$ #
The *components* allows to specify additional concentrated mass components (Force*Time^2/Length) and rotary inertias (Force*Length*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 roational DOFs are considered (D_u, D_v, D_w, R_u, R_v, and R_w).
Approximation: None.
Integration: None.
Features: None.
Loads:
Status:

2.10.2 Spring element

This element represent longitudial or torsional spring element. It is defined by two nodes, orientation and spring constant. The spring element has no mass associated, mass can be added using LumpedMass element.
Keyword: `Spring`
Parameters: `mode`$_{\text{(in)}}$ # `k`$_{\text{(rn)}}$ # `orientation`$_{\text{(ra)}}$ #
The *mode* parameter defines the type of spring element (see Table 1). The spring constant is defined by k parameter, corresponding units are [Force/Length] for longitudinal spring and [Force*Length/Radian] for torsional spring. The *orientation* defines orientation vector of spring element (of size 3) - for longitudinal

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 1: Supported spring element modes

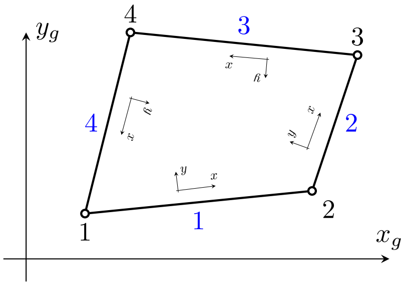

Figure 16: Quad1ht element. Node numbering, Side numbering and definition of local edge c.s.(a).

spring it defines the direction of spring, for torsional spring it defines the axis of rotation.
Note: the spring element nodes doesn't need to be coincident, but the spring orientation is always determined by *orientation* vector.

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 degrees 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).
Keyword: `Quad1ht`
Parameters: [`NIP`$_{\mathrm{(in)}}$ #]
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. (16)).

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 C_1 - 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 form cross section model. The node numbering is anti-clockwise

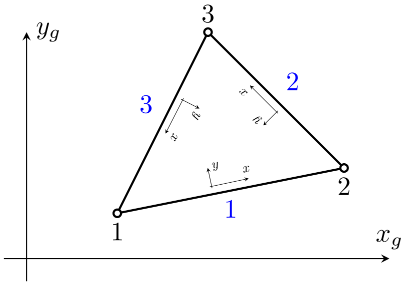

Figure 17: Tr1ht element - node and side numbering.

Keyword: `Tr1ht`
Parameters: none.
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. (17)).

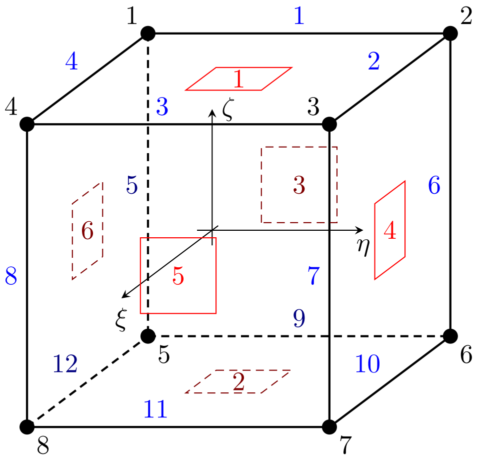

Figure 18: brick element (Node numbers in black, side numbers in blue, and surface numbers in red).

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 Brick1ht element

Represents isoparametric eight-node brick/hexahedron finite element for heat transfer problems. Each node has 1 degrees of freedom.
Keyword: `Brick1ht`
Parameters: [`NIP`$_{\mathrm{(in)}}$ #]
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. (18)).

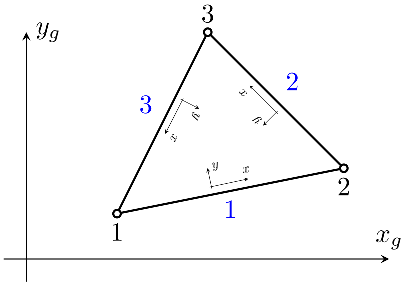

Figure 19: Tr1CBS element. Node numbering, Side numbering and definition of local edge c.s.(a).

3.3.2 Brick1hmt 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 3.3.1.

4 Elements for Fluid Dynamics problems (FM Module)

4.1 2D CBS Elements

4.1.1 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
Keyword: `Tr1CBS`
Parameters: [`bsides`$_{\rm (ia)}$ #] [`bcodes`$_{\rm (ia)}$ #]
Unknowns: Two velocity components (V_u and V_v) and pressure (P_f) are required in each node.
Boundary specification: 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.
Approximation: Linear approximation of velocity and pressure fields.
Integration: exact

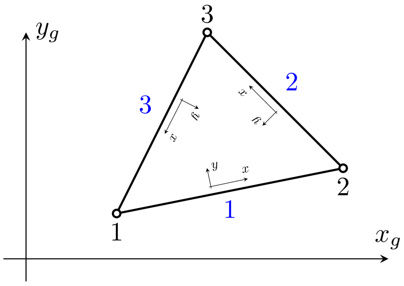

Figure 20: Tr1SUPG element. Node numbering, Side numbering and definition of local edge c.s.(a).

Loads: Constant boundary tractions are supported[1]. Body loads representing the self-weight load are supported.

4.2 2D SUPG/PSGP Elements

4.2.1 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
Keyword: `Tr1SUPG`
Parameters: [`vof`$_{(\mathrm{rn})}$ #] [`pvof`$_{(\mathrm{rn})}$ #]
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 *vof* (default is zero). Element can also be marked as allways filled with reference fluid (some form of source) using parameter *pvof* 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.

---

[1] In CBS algorithm formulation the prescribed traction boundary condition leads indirectly to pressure boundary condition in nodes associated to loaded edge. Such boundary condition is represented by PrescribedTractionPressureBC. See section on boundary conditions in OOFEM input manual.

4.2.2 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.

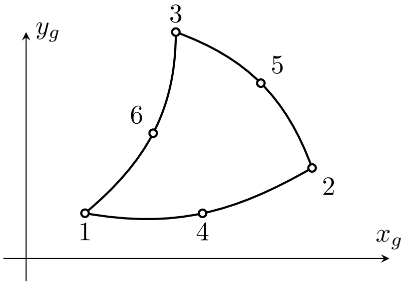

Figure 21: Tr21SUPG element - node and side numbering.

Keyword: `Tr21SUPG`
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.

4.2.3 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 ratational symmetry. The node numbering is anti-clockwise
Keyword: `Tr1SUPGAxi`
Parameters: [`vof`$_{\rm (rn)}$ #] [`pvof`$_{\rm (rn)}$ #]
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.

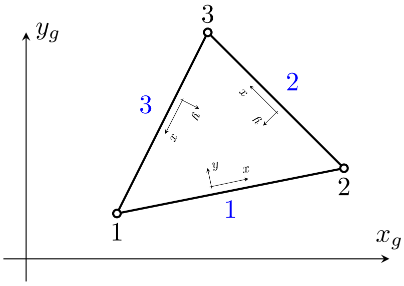

Figure 22: Tr1SUPGAxi element. Node numbering, Side numbering and definition of local edge c.s.(a).

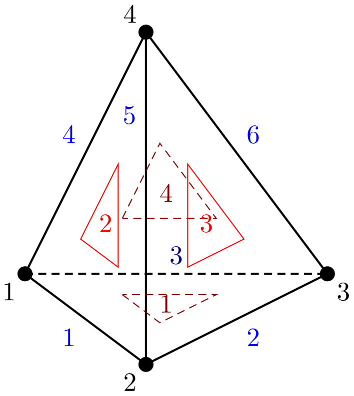

Figure 23: Tet1_3D_SUPG element.

Integration: Sevent point Gauss integration.
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 *pvof* 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.

4.3 3D SUPG/PSGP Elements

4.3.1 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).

Keyword: `TET1SUPG`
Parameters:
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 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.