CHAPTER 19. FINITE ELEMENTS LIBRARY

The finite element library (FEL) calculates stiffness, mass and stability matrices, load vectors and force (stresses) for each FE. Appropriate possible job functionals and basic functions are used here (see Chapter 18).

The basic functions depend only on the geometric characteristics of the element and the presence of bending or constrained torsion.

FEL contains:

- **By geometric characteristics**:
  - rods of constant and variable (except for physically non-linear) sections: beam, Taurus, I-beam, channel bar, RHS, ring;
  - 3- and 6-knot triangles, 4- and 8-knot quadrilaterals of constant thickness, additional knots in the midpoints of the sides;
  - 8- and 20-knot hexahedrons, 6- and 15-knot pentahedrons, 4- and 10-knot tetrahedrons, 5- and 13-knot tetrahedral pyramids, additional nodes in the middle of edges;
  - single node elements.
- **By types of tasks**:
  - linear elements;
  - non-linear elastic bending elements;
  - geometrically non-linear elements;
  - non-linear elastic geometrically non-linear elements;
  - elastic-plastic elements;
  - soil elements;
  - special elements.

All the finite elements, except for single-node and some special ones, have a local right Cartesian coordinate system X1, Y1, Z1.

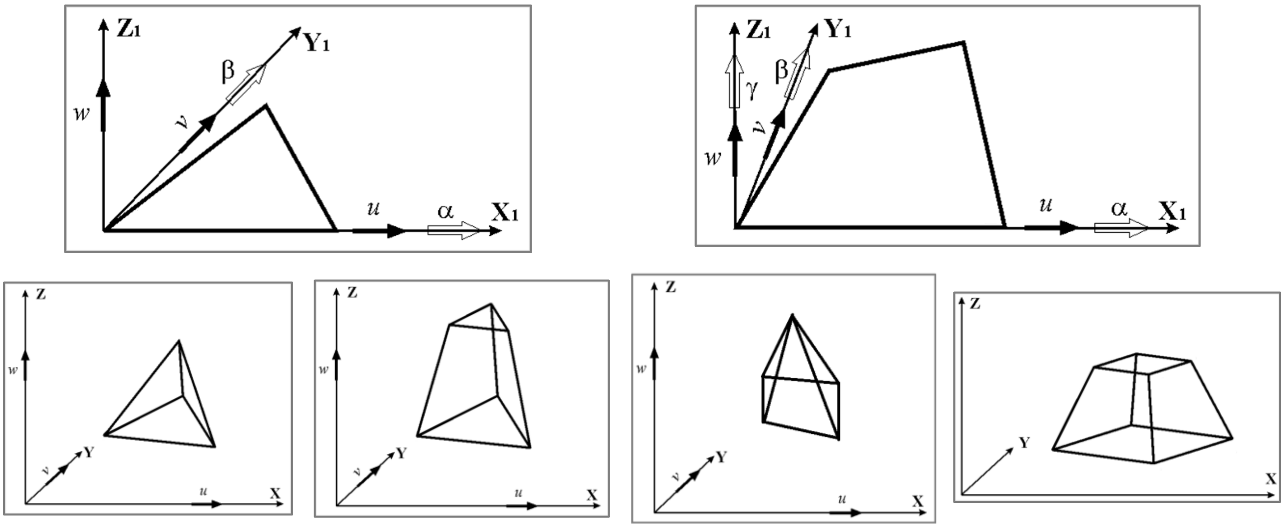

Fig. 19.1. Lamellar and volumetric FE (except for special ones) coordinate system

There are no curvilinear elements of rods and shells in FEL, because the convergence of rectilinear and plane elements was proved in [18.21, 18.22, 18.59, 18.77].

Nodal unknowns (degrees of freedom) of the elements correspond to the type of problem being created and are X, Y, Z displacements and UX, UY, UZ rotations in the local coordinate system. The bar element with constrained torsion has the seventh degree of freedom W (considering the deformation of the section). For two-dimensional elements, an additional nodal unknown is allowed, namely, the rotation around an axis perpendicular to the plane [18.90], which allows to get rid of geometric variability.

Local loads are specified in the general or local coordinate systems in the directions corresponding to the nodal unknowns of the element.

19.1 BARS

Local axes

The X1 axis is directed from the first node to the second. For spatial systems, the Y1 and Z1 axes are the main central axes of inertia.

By default, it is assumed that for arbitrarily oriented rods, the Z1 axis is always directed to the upper half-space, and the Y1 axis is parallel to the horizontal XOY plane of the global coordinate system, while for vertical rods, it is parallel to the Y axis of the global coordinate system and directed in the opposite direction. If the position of the main central axes of the real bar does not coincide with the default position, then it is necessary to set the position of the Y1 axis (Fig. 19.2).

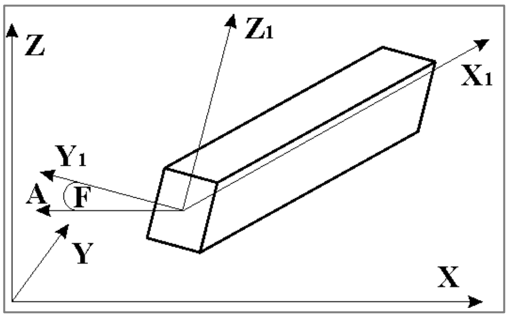

Fig. 19.2. Universal FE of a spatial bar

Local Axes of node of the equivalent bar (FE type No. 110) is similar to the general FE of the spatial bar (FE 10). The equivalent bar is not involved in the finite element calculation. This element allows collecting of forces from linear two-dimensional and linear three-dimensional elements, in order to further select and check the design according to the selected standards. For an equivalent bar, the force collection includes those nodes that are located between the normal planes passing through two nodes of the equivalent bar.

Sections

Beam, Taurus, I-beam, channel bar, RHS (rectangular hollow section), ring; of constant and variable (except physically non-linear) sections, for physically linear sections, like a corner, a cross, non-symmetrical Taurus and I-beam of constant section.

Assign Rigid Inserts and Hingers are allowed at the beginning and end of the flex section.

For geometrically linear bending bars, shear can be taken into account. To do this, the appropriate parameter should be set.

Elastic foundation (18.9.20) in linear and elastoplastic elements can also work as one-sided.

The stiffness matrix is built for the flexible part of the bar.

Element loads

Concentrated load; evenly distributed; trapezoid; temperature - uniform heating or cooling, etc.

Bindings of concentrated and trapezoidal loads are set relative to the flexible part of the bar, i.e. possible negative bindings. A uniformly distributed load can also be applied to rigid inserts.

For concentrated and uniformly distributed loads, the bindings, relative to the center of the section can be set. In this case, additional moment loads are automatically calculated. These bindings are also taken into account when constructing the stability matrix in accordance with (18.9.25). For a bar with constrained torsion, the position of the center of torsion is taken into account, which, according to (18.9.23), gives an additional torque.

Loads are calculated in the coordinate system of the flexible part, for non-linear bars at the beginning and at the end of the flexible part, for linear bars, you can set the number of sections in which forces are calculated**.**

T Bar types, nodal unknowns, calculated forces:

- Spatial truss (X, Y, Z; NX);
- beam (Z, UX, UY; MX, MY, QZ);
- the spatial frame excluding warping of bars (X, Y, Z, UX, UY, UZ; NX, NY, NZ, MX, MY, MZ);
- the spatial frame considering warping of bars (X, Y, Z, UX, UY, UZ, W; NX, NY, NZ, MX, MY, MZ, MW);
- Equivalent bar element (NX, NY, NZ, MX, MY, MZ).

In case the is an elastic foundation, the corresponding reactions RY, RZ are calculated.
Truss elements and beams are special cases of a spatial frame. Beam elements are linear only.

19.2 Two-dimensional elements

Local axes system

The X1 axis is directed from the first node to the second. The Y1 axis lies in the plane passing through the first three nodes and forms an acute angle with the vector directed from the first node to the third.

It is possible to set **axes of stress alignment**. For orthotropic elements, **axes orthotropy** are specified. These coordinate systems are obtained by rotating the local system around the Z1 axis. Forces are calculated at the center of gravity of the element.

Local Axes of the equivalent shell (FE №142, 144-type) is similar to the FE thin shell (FE 42, 44). The equivalent shell is not involved in the finite element calculation. This element allows you to collect forces from linear three-dimensional elements, with the aim of further Proportioning and Checking of structures according to the selected standards. For an equivalent shell, the forces are collected from the stresses of the volumes of the plane of the faces that are intersected by the normal drawn from the center of the equivalent shell, while the projection of the face falls into the body of the equivalent shell.

For geometrically linear bending slabs and plates, shear can be taken into account. To do this, unlike rods, the element type should be changed.

Elastic foundation (18.9.13) in linear and elastoplastic elements can also work as compression only. With the elastic base, the reaction RZ is calculated.

For plates, **rigid inserts** are allowed in nodes having the same length and being perpendicular to the plane of the element.

.

Element loads

Concentrated force; uniformly or trapezoidal distributed, distributed at an element or along the line; by temperature — uniform heating or cooling, etc.

Types of two-dimensional elements, nodal unknowns, calculated forces (stresses NX, NY, NXY):

- bending plate (Z, UX, UY; MX, MY, MXY, QX, QY);
- plane stress and plane strain, vertical stresses (X, Z; NX, NZ, NXZ) and arbitrary position (X, Y, Z; NX, NY, NXY);
- plate (X, Y, Z, UX, UY, UZ; NX, NY, NXY, MX, MY, MXY, QX, QY);
- equivalent insert (NX, NY, NXY, MX, MY, MXY, QX, QY).

19.3 SOLID ELEMENTS

Local Axes of elements is built in the same way as for two-dimensional elements. It is possible to set axes of stress alignment. For the orthotropic elements, **axes of orthotropy** are specified.

Loads on the element:

- concentrated force;
- uniformly distributed along the element or on face;
- temperature — uniform heating or cooling.

Nodal unknowns, calculated stresses:

- X, Y, Z;
- NX, NY, NZ, NXY, NXZ, NYZ.

Stresses are calculated at the center of gravity of the element.

The above described is common to both linear and non-linear elements.

19.4 Geometrically nonlinear elements

They are applied when it is necessary to take into account changes in the geometry of the structure. The stiffness matrix is formed in the coordinate system of the deformed position, the corresponding stability matrix (18.9.25) for bars, (18.9.18) for plates or (18.9.7) for three-dimensional elements is added to it, the total matrix is then converted to the original coordinate system using a special matrix of cosines, which is built from the total displacements.

The forces are calculated in the coordinate system of the deformed position.

Types of geometrically nonlinear elements:

- strings and membranes;
- rods and plates, von Karman's theory;
- rods and plates, tight bend;
- rod taking into account constrained torsion, tight bend;
- solid elements.

19.5 Physically nonlinear elements

They are applied when it is necessary to take into account the nonlinear relationship between deformations and stresses. It is possible to have two materials (concrete and reinforcement), for which the following dependencies are specified:

- Exponentially:

$$\sigma = E\varepsilon \qquad \text{at } \varepsilon <= \varepsilon_u = \sigma_u/E;$$
$$\sigma = (R - \sigma_u) \cdot (1 - exp(-\frac{E(\varepsilon - \varepsilon_u)}{R - \sigma_u})) + \sigma_u \qquad \text{at } \varepsilon > \varepsilon_u = \sigma_u/E)); \qquad (19.5.1)$$

where *R* is the ultimate strength;

σ_u is the elastic limit.

For reinforced elements, two types of exponential laws can be used as the main filler: "11 - exponentially dependent material" and "15 - exponentially dependent concrete". Their difference is that in the first case, the reinforcement and concrete work together, and in the second, the reinforcement slippage or concrete collapse is taken into account.

- Trilinear (complies with construction regulations):

$$\sigma = E\varepsilon \qquad \text{at } \varepsilon < \varepsilon_u = \sigma_u/E)),$$
$$\sigma = \sigma_u + \varepsilon * (R - \sigma_u)/(\varepsilon_0 - \varepsilon_u) \qquad \text{at } \varepsilon_u < \varepsilon < \varepsilon_0,$$
$$\sigma = R \qquad \text{at } \varepsilon > \varepsilon_0; \qquad (19.5.2)$$

at $\sigma_u = R$ the three-linear dependence turns into the two-linear one.

- Piecewise-linear description is specified by the values $\{\varepsilon_n, \sigma_n\}$, with $\{\varepsilon_n < \varepsilon_{n+1}\}$,$\{\sigma_n < \sigma_{n+1}\}$;

- Piecewise-linear description is specified by the values $\{\varepsilon_n, \sigma_n\}$, with $\{\varepsilon_n < \varepsilon_{n+1}\}$,$\{\sigma_n < \sigma_{n+1}\}$;

for the base material (concrete) of elastoplastic two-dimensional and three-dimensional elements, **Geniev strength state condition** is applied:

$$-(R_c + R_s)S_1 + S_2^2 + R_c * R_s < 0, \quad (19.5.3)$$

$\sigma_1 \geq \sigma_2 \geq \sigma_3$ — stresses,

$S_1 = \sigma_1 + \sigma_2 + \sigma_3,$

$S_2 = \sqrt{\frac{(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2}{2}}.$

For dependencies (19.5.1), (19.5.2), $R = R_c$ at $\varepsilon < 0$ и $R = R_s$ at $\varepsilon < 0$, R_c and R_s should be taken as the ultimate strength in uniaxial compression and tension.

Axes of Stress Alignment of 2D and 3D elements coincide with the reinforcement axes, and the forces (stresses) are calculated in these axes. Stress in non-bending elements (2D and 3D) is calculated separately for each of the two materials.

For bending bars, section fragmentation into primitive rectangles is specified.

Reinforcement inclusions:

For bending bars, the areas of reinforcing bars and their position in the section are specified. For bendable plates, reinforcing inclusions are specified in two directions (area and position in the section). For non-bending elements, the percentage of reinforcement in each direction is indicated.

All physically non-linear elements can be calculated on creep. It is required to specify the law of decreasing the modulus of elasticity in time, according to the Eurocode or to Piecewise-linear description.

Types of physically nonlinear elements:

- non-linearly elastic elements: rods and plates;
- elastoplastic: rods, plates, elements of plane stress state and plane deformation, of vertical and arbitrary positions, three-dimensional elements.

19.6 Physically and Geometrically nonlinear elements

They are applied if it is necessary to simultaneously take into account physical and geometric nonlinearities. FEL contains non-linearly elastic rods and tight bend plates.

19.7 Soils

They are used to calculate soil massifs and to solve filtration problems.

Consideration of soil specifics is ensured by setting one of the strength conditions:

- Botkin condition:

$$sin\,\phi\,\sigma_0 + \sigma_i - 2C\,cos\,\varphi \leq 0; \quad (19.7.1)$$

- Drucker-Prager condition:

$$2\,sin\,\phi\,\sigma_0 + (3 - sin\,\phi)\sigma_i - 6C\,cos\,\phi \leq 0; \quad (19.7.2)$$

- Coulomb-Mohr condition:

$$sin\,\phi\,(\sigma_1 + \sigma_3) + (\sigma_1 - \sigma_3) - 2C\,cos\,\phi \leq 0; \quad (19.7.3)$$

- Analytical failure theory:

$$-((\sigma_1+\sigma_3)/s\gamma+1)^{\alpha}((\sigma_1+\sigma_3)/s\delta+1)^{\beta}+(\sigma_1-\sigma_3)/\tau \le 0, \qquad (19.7.4)$$

where $\sigma_0 = \sigma_1+\sigma_2+\sigma_3$;

$\sigma_i = \sqrt{\frac{(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2}{2}}$;

$\sigma_3 \le \sigma_2 \le \sigma_1 \le R_s$ — principal stress;

R_s — limit tensile stress;

C — adhesion

ϕ — angle of internal friction.

It is allowed to take into account the initial soil stress factor. The quantities included in (19.7.4) are determined experimentally.

FEL contains 2D and 3D soil elements. Two-dimensional elements are vertical and operate under plane deformation conditions.

19.8 Special elements

Elements of the elastic pile (type No. 57) simulate the operation of a pile. They can be one- and two-nodal.

Non-reflecting boundary elements (types No. 62, 63, 64, 65, 68) simulate non-reflecting conditions at the boundary of a selected area in dynamic calculations of soil masses. In a static problem, bonds are simulated along a direction, perpendicular to the surface.

FEL contains 1D and 2D non-reflective border elements.

In the dynamics on the boundary where such elements are located, the following conditions are satisfied:

$$\sigma_n = c_n \cdot \rho \cdot u'_n; \qquad (19.7.5)$$

$$\sigma_\tau = c_\tau \cdot \rho \cdot u'_\tau; \qquad (19.7.6)$$

where σ_n is the normal stress on the boundary acting perpendicular to the boundary;

σ_τ is shear stress on the boundary in the plane of the boundary;

ρ is the soil density;

u'_n, u'_τ — are the speed of soil at the boundary, respectively, perpendicular to the boundary and tangentially;

c_n and c_τ are the velocities of the longitudinal wave (P-wave) and transverse wave, respectively (S-waves) in the soil at the boundary. The velocities of longitudinal and transverse waves in the soil can be obtained from the dependencies:

$$c_n = \sqrt{\frac{E\cdot(1-\nu)}{\rho\cdot(1+\nu)\cdot(1-2\nu)}};$$

$$c_\tau = \sqrt{\frac{E}{2\rho\cdot(1+\nu)}} = \sqrt{\frac{G}{\rho}};$$

where E is the soil elasticity modulus at the boundary;

ν is the Poisson's ratio of the soil at the boundary;

G is the soil shear modulus at the boundary.

Elements of peripheral elastic foundation (can also act as one-sided) have one nodal unknown W, defined relative to the common coordinate system. As a result of the calculation, the reactions at the nodes of the element are calculated.

A two-node element (FE 53) models the repulsion of a strip of soil lying outside the slab and perpendicular to its contour (Fig. 19.3). The bedding coefficients C1 and C2 of the Pasternak model are specified for it.

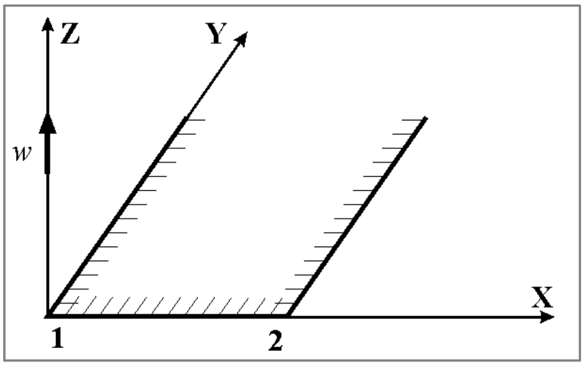

Fig. 19.3. Contouring two-node elastic foundation FE

Single-noded element (FE 54) models the reaction of the soil corner zone, adjacent to the corner of the slab (Fig. 19.4). For it, the bed coefficient C2 of the Pasternak model and the angle of the soil zone in degrees, measured between the normals to the sides included in the corner of the slab, are specified.

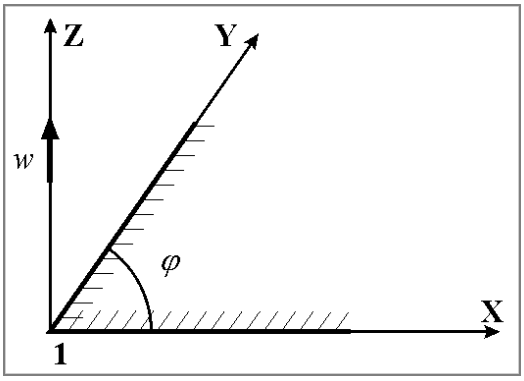

Fig. 19.4. Contouring single-node elastic foundation FE

Elements of elastic link model elastic links in a node (single-node FE 56) or between adjacent nodes (two-node FE 55). The nodal unknowns are defined with respect to the axes of the global or local coordinate system. Nodes FE 55 may have the same coordinates.

Forces are reactions at nodes.

In Cross Sections/Stiffness Editor the FE stiffness parameters are to be set:

- Axial stiffness of link under tension-compression along global axes Rx, Ry, Rz;
- Rotational stiffness of link under rotation around global axes Rux, Ruy, Ruz.

The length of the element in the formation of the stiffness matrix is taken equal to one.

The two-node FE 55 does not have a local coordinate system.

One-sided pre-tensile elements simulate pre-stressing (FE 207) or tensioning (FE 208). In this case, the calculation is performed by the step-iterative method.

The stiffness is specified numerically in the **Special sections** of the section/stiffness editor by entering a value in the text string. It is also necessary to set the compressive or tensile force ***Nmax***.

For the first loading of a circuit containing FE 207 or FE 208 it is recommended to specify stressing or tension. Subsequent loadings are linked to the first. All loadings, except for the first one, can contain any loads, like dead weight, temperature, etc.

Attention! *If at any calculation step the value of Nmax is exceeded in at least one of the FEs, the calculation will stop, and the elements will retain the efforts of the previous step. For this reason, it is recommended to specify at least two sample times.*

Geometrically non-linear special pre-tensile FE models pretension in geometrically non-linear applications. Stiffnesses are set as for a regular bar. In the absence of pretension, it works as an element of the thread.

Special FE of inelastic links considering limit stresses model inelastic links in a node (FE 256) or between two nodes (FE 255).

The elements make it possible to take into account unequal limiting forces, for example, in tension and compression.

The nodes of element 255 cannot have the same coordinates. The length of the element in the formation of the stiffness matrix is taken equal to one.

In the section/stiffness editor, the FE stiffness parameters are set:

- linear stiffness of the connection for tension-compression and rotations;
- ultimate tension-compression forces and rotations.

One-sided friction elements model friction in a one-sided connection. For a single node FE 263, the link is oriented along one of the node's global or local coordinate axes. For a two-node FE 264, the connection direction is built in accordance with the coordinates of the nodes that describe the element, and coincides with the longitudinal axis X1.

The Coulomb friction condition is applied: $|\tau| \leq -\gamma\sigma$,

where τ и σ are shear and normal stresses;

γ is the static friction coefficient.

In the section/stiffness editor, stiffness parameters are set:

- R — Axial stiffness of link under tension-compression, tf/m;
- Q — in length stiffness for adhesion, tf/m;
- γ — coefficient of static friction;
- b — gap, mm.

The switch indicates the work of the connection either in tension or in compression. Also, using the drop-down box for a single-node FE, the direction of communication is set.

The nodes of element 264 cannot have the same coordinates.

The length of the element in the formation of the stiffness matrix is taken equal to one. The clearance value with a «+» sign in tension or with a «–» in compression.

Elements of a one-way connection, considering the signs of displacements, model one-way connections in a node (one-node FE 256) or between two nodes (two-node FE 255); being the analogues of FE 56 and FE 55, taking into account one-sided operation.

The nodes of element 255 cannot have the same coordinates. The length of the element in the formation of the stiffness matrix is taken equal to one.

In the section/stiffness editor, the following must be specified:

- linear stiffness of the connection for tension-compression and rotations;
- clearances in all directions.

Using the switch, the principle of operation must be specified:

- elastic under tension;
- elastic under compression;
- elastic link.

Elements for calculating the stiffness characteristics of sections

FEL contains one-dimensional elements for thin-walled section (type №103) and two-dimensional elements for massive section (type №112 - triangle, and type №119 - quadrilateral).

Heat exchange elements

FEL contains 1D and 2D elements. They are used in the heat conduction problem for modeling the Newton-Richmann boundary condition.

$$q = \alpha \cdot (T - T_{medium}); \qquad (19.7.7)$$

where q is the power of the heat flow through a platform of unit area at the boundary of the medium;

α is the coefficient of surface heat transfer;

T is the body temperature at the boundary of the medium;

T_{medium} is the temperature of the external medium at the boundary.

Soil elements of interface

This FE makes it possible to take into account the incompatibility of deformations of the base and the structure at the boundary of their contact. For solving plane problems, a rectangular physically nonlinear plate FE of the interface is implemented, and for volumetric physically nonlinear solid FE of interface (spatial triangular and quadrangular prisms).

To describe the deformation parameters of the interface elements, in addition to the parameters describing the operation of the adjacent soil element, virtual interface thickness (along the Z1 axis) - Hf, as well as interface strength factor Riner (in the range 0 - 1) are specified. For planar elements, the local axis X1 FE of the interface must be parallel to the edge of the adjoining structural element, and for solid elements, the X1Y1 FE interface plane must be parallel to the face of the adjoining structural element.

The implementation used the theory set out in the manual [18.63].

$$E_r = 2 \cdot G_r \cdot \frac{1-\gamma_r}{2 \cdot \gamma_r}; \ \gamma_r = 0.45; G_r = Riner^2 \cdot G_{soil};$$

$$|\tau| \le Riner \cdot (C_r - \sigma \cdot tan(\phi)); \qquad (19.7.8)$$

$$\sigma \le Riner \cdot R_t; \qquad (19.7.9)$$

where E_r is the elastic modulus of the interface;

G_r is the interface shift modulus;

Riner is the interface strength reduction factor;

G_{soil} is the soil base shear modulus;

τ, σ are shear and normal stresses, respectively;

φ is the internal friction angle;

C_r is the cohesion of interface;

R_t is the axial tensile strength of the soil.

Plate of elastic link

Plate (FE 94, 96, 97, 98) is intended for modeling linear joints of panel buildings. Each of the FE nodes has six degrees of freedom: X, Y, Z, UX, UY, and UZ. The stiffnesses of such FEs are specified in the global coordinate system. When forming the stiffness matrix, the width of the element along the orthotropy axis Y1 is assumed to be equal to one (it is believed that the orthotropy axis Y1 should be directed across the joint). The forces are calculated in the coordination axes.

Plate of inelastic link

Plate (FE 294, 296, 297, 298) is intended for modeling linear and non-linear joints of panel buildings. Each of the FE nodes has six degrees of freedom: X, Y, Z, UX, UY, and UZ. The stiffnesses of such FEs are specified in the global coordinate system. When forming the stiffness matrix, the width of the element along the orthotropy axis Y1 is assumed to be equal to one (it is believed that the orthotropy axis Y1 should be directed across the joint). The forces are calculated in the coordination axes.

Elements of filtration shielding layer

One-dimensional elements are designed to model the drainage-only flow boundary condition when solving a flat problem, and flat elements when solving a spatial filtration problem. A flat FE can be represented as a triangular (172) or quadrilateral (174) element. The nodal unknown in this type of finite element is pressure.

Drainage-only flow boundary condition can be written as an equation:

$$V_n = R_0 \cdot p; \tag{19.7.10}$$

where V_n is the component of the pore fluid velocity in the direction of the outward normal to the surface;

R_0 is the seepage coefficient;

p is the current pore pressure at this point on the surface.

✍ *For modeling the drainage-only flow boundary condition, it is recommended to take a seepage coefficient of about* $10^5 \cdot K_f/(\rho \cdot c)$*, where* K_f *is the permeability of the underlying material;* ρ *– is the specific gravity of the liquid;* c *– is the size of the soil element in the direction of the thickness of the base soil layer near the surface.*

19.9 FEL Composition

- **Bars**:
  - Linear:
    - №4 — spatial truss;
    - №3 — beam rod;
    - №10 — spatial frame without constrained torsion;
    - №7 — spatial rod with warping torsion;
    - №110 – equivalent rod.
  - Physically non-linear, non-linear elasticity:
    - №204 — spatial truss;
    - №210 — spatial frame.
  - Geometrically non-linear:
    - №304 — spatial thread;
    - №310 — spatial frame;
    - №309 — strong bend without constrained torsion;
    - №307 — strong bend with constrained torsion.
  - Physically and geometrically non-linear, non-linear elasticity:
    - №404 — spatial thread;
    - №410 — spatial frame, strong bend.
  - Physically non-linear, elastoplasticity:
    - №504 — spatial thrust;
    - №510 — spatial frame.
- **2D elements**:
  - Linear elements of a bending plate.
  - Subtle, Kirchhoff theory:
    - №12 — a three-nodal triangle;
    - №13 — a six-node triangle;
    - №19 — a four-node quadrilateral;
    - №20 — an eight-node quadrilateral.
  - Thick, Reisner's theory:
    - №16 — a three-nodal triangle;
    - №14 — a six-node triangle;
    - №17 — four-node quadrilateral;
    - №18 — an eight-node quadrilateral.
  - Linear elements of plane stress and plane strain:
    - Vertical:
      - №22 — three-node triangle;
      - №25 — a six-node triangle;
      - №30 — four-node quadrilateral;
      - №29 — an eight-node quadrilateral.
    - Arbitrary position:
      - №24 — a three-node triangle;
      - №26 — a six-node triangle;
      - №27 — a four-node quadrilateral;

        - №28 — an eight-node quadrilateral.
- Linear plated elements:
    - Subtle, Kirchhoff theory:
        - №42 — a three-node triangle;
        - №43 — a six-node triangle;
        - №44 — a four-node quadrilateral;
        - №50 — an eight-node quadrilateral.
    - Thick, Reisner's theory:
        - №46 — a three-node triangle;
        - №48 — a six-node triangle;
        - №47 — a four-node quadrilateral;
        - №49 — an eight-node quadrilateral.
- Shell equivalent elements:
    - №142 — a three-node triangle;
    - №144 — a four-node quadrilateral.
- Physically nonlinear elements of plane stress and plane strain, elastoplasticity:
    - Vertical:
        - № 222 — a three-nodal triangle;
        - №225 — a six-node triangle;
        - № 230 — a four-node quadrilateral;
        - № 229 — an eight-node quadrilateral.
    - Arbitrary position:
        - №224 — a three-node triangle;
        - №226 — a six-node triangle;
        - №227 — a four-node quadrilateral;
        - №228 — an eight-node quadrilateral.
- Soil body elements, vertical, planar deformation:
    - №282 — a three- node triangle;
    - №283 — a six-node triangle;
    - №284 — a four-node quadrilateral;
    - №285 — an eight-node quadrilateral.
- Physical non-linear elements of shells, non-linear elasticity:
    - Subtle, Kirchhoff theory:
        - №242 — a three-node triangle;
        - №243 — a six-node triangle;
        - №244 — a four-node quadrilateral;
        - №250 — an eight-node quadrilateral.
    - Thick, Reisner's theory:
        - №246 — a three- node triangle;
        - №248 — a six-node triangle;
        - №247 — a four-node quadrilateral;
        - №249 — an eight-node quadrilateral.
- Geometrically non-linear elements of thin shells (an additional attribute is set: Von Karman theory, membrane, strong bending):
    - №342 — three-nodal triangle;
    - №343 — a six-node triangle;

        - №344 — a four-node quadrilateral;
        - №350 — an eight-node quadrilateral.
    - Physical nonlinear elements of shells, elastoplasticity:
        - №442 — a three- node triangle;
        - №443 — a six-node triangle;
        - №444 — a four-node quadrilateral;
        - №450 — an eight-node quadrilateral.
    - Physical nonlinear elements of shells, elastoplasticity:
        - Subtle, Kirchhoff theory:
            - №542 — a three-node triangle;
            - №543 — a six-node triangle;
            - №544 — a four-node quadrilateral;
            - №550 — an eight-node quadrilateral.
        - Thick, Reisner's theory:
            - №546 — a three-node triangle;
            - №548 — a six-node triangle;
            - №547 — a four-node quadrilateral;
            - №549 — an eight-node quadrilateral.
- **3D solid elements**:
    - Linear:
        - №36 — an eight-node hexagon;
        - №37 — a twenty- node hexagon;
        - №34 — a six-node pentahedron;
        - №39 — a fifteen-node pentahedron;
        - №32 — a four-node tetrahedron;
        - №38 — a ten-node tetrahedron;
        - №35 — a five- node tetrahedral pyramid;
        - №40 — a thirteen node tetrahedral pyramid.
    - Physically non-linear, elastoplasticity:
        - №236 — eight-node hexagon;
        - №237 — twenty-node hexagon;
        - №234 — six-node pentahedron;
        - №239 — fifteen-node pentahedron;
        - №232 — a four-node tetrahedron;
        - №238 — ten-node tetrahedron;
        - №235 — a five-node tetrahedral pyramid;
        - №240 — a thirteen-node tetrahedral pyramid.
    - Elements of the soil body:
        - №276 — eight-node hexagon;
        - №277 — twenty-node hexagon;
        - №274 — six-node pentahedron;
        - №279 — fifteen-node pentahedron;
        - №272 — four-node tetrahedron;
        - №278 — ten-node tetrahedron;
        - №275 — a five-node tetrahedral pyramid;

    - №280 — thirteen-node tetrahedral pyramid.

- **Special elements**:
  - Одно- и двухузловые элементы упругой связи, №56, №55;
  - One- and two-node elements of elastic link, №56, №55;
  - One- and two-node elements of elastic pile №57;
  - One- and two-node elements of the contour elastic base, possibly one-sided, №54, №53;
  - Geometrically nonlinear special pre-tensile FE, №308;
  - One-sided pre-tensile elements, №208 — tension, №207 — compression;
  - One- and two-node elements of one-side link, №266, №265 — by displacement, №256, №255 — by strain;
  - Plate linear FE of elastic link:
    - №94 — triangular;
    - №96 — triangular, three-node (if FE are used with intermediate nodes)
    - №97 — quadrangular, four-node;
    - №98 — quadrangular, four-node (if you use FE with intermediate nodes).
  - Physically non-linear plate FE of inelastic link:
    - №294 — triangular, three-node;
    - №296 — triangular, three-node (if FE is used with intermediate nodes);
    - №297 — quadrangular, four-node;
    - №298 — quadrangular, four-node (if FE is used with intermediate nodes).
  - One- and two-node elements of a one-sided friction, Coulomb’s law, №263, №264;
  - Physically nonlinear soil interface elements:
    - №268 — plated, four-node;
    - №269 — spatial three- or quadrangular prism.
  - Elements of filtration shielding layer:
    - №178 — one-dimensional;
    - №172 — a three-node triangle;
    - №173 — a three- node triangle (if FE with intermediate nodes are used);
    - №174 — our-node quadrilateral;
    - №175 — our-node quadrilateral (if FE with intermediate nodes are used).
  - Non-reflective border elements:
    - one-dimensional №68;
    - two-dimensional:
      - №62 — a three-node triangle;
      - №63 — a six-node triangle;
      - №64 — a four-node quadrangle;
      - №65 — an eight-node quadrilateral.
  - Elements for calculating the stiffness characteristics of sections:
    - one-dimensional for thin-walled section №103;
    - two-dimensional:
      - №112 — a three-node triangle;
      - №113 — a six-node triangle;
      - №119 — fa our-node quadrupole;
      - №120 — an eight-node quadrilateral.

- Elements of surface heat transfer:
  - single node №151;
  - one-dimensional №168;
  - two-dimensional:
    - №162 — a three-node triangle;
    - №163 — a six-node triangle;
    - №164 — a four-node quadrupole;
    - №165 — an eight-node quadrilateral.