
Version
April 2018

Program

# RFEM 5

Spatial Models Calculated According to
Finite Element Method

## Theory




Am Zellweg 2
93464 Tiefenbach
Germany

Tel.: +49 9673 9203-0
Fax: +49 9673 9203-51
E-mail: info@dlubal.com
Web: www.dlubal.com

# Contents

# 1 Calculation

RFEM uses the finite element method based on the updated Lagrangian formulation and the displacement formulation.

## 1.1 Nonlinearity

Nature is nonlinear. Although we often look to simplify the world around us, with the increase of hardware capacity and software improvements in the last decades the linearization of real life engineering problems is not always necessary. RFEM offers a huge variety of nonlinear models covering many areas of use.

## 1.2 Parallelization

In order to achieve maximum computational efficiency, calculation is parallelized on different levels – the direct Cholesky linear solver is parallelized by parallel elimination of matrix rows, the dynamic relaxation method is parallelized through its natural explicit definition. Many other blocks are also parallelized (local element matrix assembly or beam cross-sectional mesh generation).

## 1.3 Types of Analysis

The types of deformation analysis available in RFEM can be divided into:

- Static analysis
- Dynamic analysis
- Stability analysis

## 1.4 Static Analysis

Nonlinearities in RFEM can be divided into the following:

- Structural nonlinearity (nonlinear releases (= hinges), nonlinear supports, etc.)
- Material nonlinearity (nonlinear elasticity, plasticity, masonry calculation, calculation of cables and membranes, etc.)
- Geometrical nonlinearity

Geometrical nonlinearity is handled in RFEM according to the following user settings:

- Geometrically linear static analysis (small deformation theory, the so-called *first-order theory*)
- The *second-order theory* (P-delta analysis, improved small deformation theory taking axial forces into account)
- Large deformation theory (large deformations and large strains, nonlinear methods including the Newton-Raphson method, the Picard secant method and their combination, the so-called *third-order theory*)
- Post-critical analysis (large deformations and large strains, nonlinear methods including the modified Newton method and the dynamic relaxation method)

The large deformation theory and post-critical analysis differ only in the nonlinear methods used. The latter choice allows the calculation of post-critical behavior of a structure which goes through a singular point, i.e., where the stiffness matrix becomes singular. For a detailed description of the nonlinear methods, see section *Nonlinear Solvers* below. In case of the large deformation theory, the axial strain is computed with respect to the actual and not reference length, as in the first-order theory. Under a sufficient number of load steps, the axial strain numerically tends to a logarithmic strain definition. In case of beams computed under the large deformation theory, the axial stiffness is computed directly according to the logarithmic strain definition, which gets precise results with one load increment only.

### 1.4.1 Linear Solvers

The linear solvers available in RFEM are:

- Direct linear solver – parallelized Cholesky solver for symmetric sparse matrices (default choice)
- Iterative linear solver for symmetric sparse matrices

The first method is faster except for large positions, where the iterative approach can be less time demanding. The iterative linear solver, on the other hand, can be easier parallelized. The choice between the nonlinear and the linear solver is up to the user.

### 1.4.2 Nonlinear Solvers

Nonlinear calculations in general yield a system of nonlinear algebraic equations, which need to be solved. The robustness of the nonlinear solver is a crucial part of the calculation process within the framework of finite element analysis. The nonlinear method transforms the nonlinear problem into a sequence of linear problems, which are then solved by a linear solver. The nonlinear solvers available in RFEM are:

- Newton–Raphson method (default choice)
- Newton–Raphson combined with the Picard method
- Picard method (secant method)
- Newton–Raphson with constant stiffness matrix
- Modified Newton–Raphson method
- Dynamic relaxation

The Newton–Raphson nonlinear method is preferred in case of a continuous right-hand side. In case of discontinuities, the Picard method can be used (especially in combination with the Newton–Raphson as a corrector) as a more robust choice. The post-critical behavior, where the solver has to overcome limit points with singular stiffness matrices, is either solved by the modified Newton–Raphson or by the dynamic relaxation method.

#### 1.4.2.1 Newton–Raphson

In this method, the tangential stiffness matrix is calculated as a function of the current deformation state and inverted in every iteration cycle. In the majority of cases, the method features a fast (quadratic) convergence.

#### 1.4.2.2 Picard

This method is also known as the *fixed-point iteration method*. It can be thought of as a finite-difference approximation of the Newton method. The difference is considered between the current iteration cycle and the initial iteration cycle in the current load step. The method does not converge as quickly as Newton's method in general, but it can be more robust for some nonlinear problems.

### 1.4.2.3 Newton–Raphson Combined with Picard

The analysis starts as the Picard method and then switches to the Newton method. The idea behind this is to use the robust method far from equilibrium and the fast convergent method near equilibrium. The first *n* percent of iterations, for which the Picard method is used, are set in the *Calculation Parameters* dialog box.

### 1.4.2.4 Newton–Raphson with Constant Stiffness Matrix

This method is like the Newton–Raphson method. The difference is that the stiffness matrix is assembled only once in the first iteration cycle. It is then used in all subsequent cycles. Therefore, this method is faster, but not as robust as the Newton-Raphson method and the modified Newton–Raphson method.

### 1.4.2.5 Modified Newton–Raphson

The method is used for post-critical analysis, that is, for problems in which a region of instability has to be overcome to solve the problem. In the case of instability, where the stiffness matrix cannot be inverted, the stiffness matrix from the last stable iteration step is used. This matrix is used until the stability region is reached again. The methods enables the handling of decay diagrams:

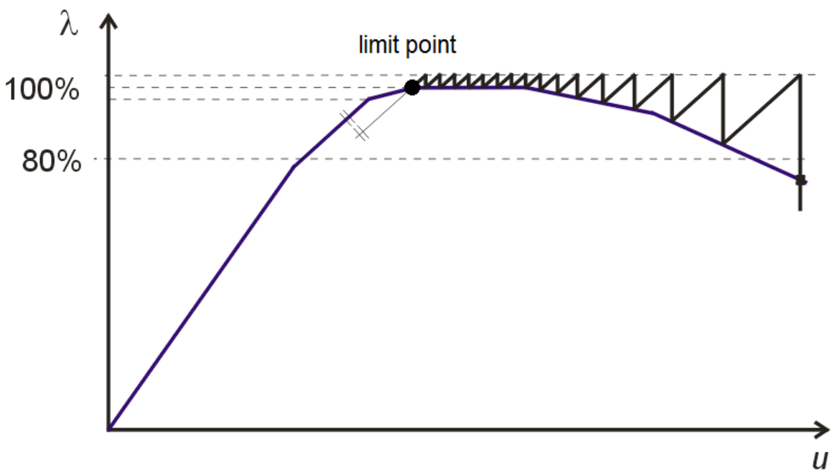



Figure 1.1: Overcoming limit point by means of the modified Newton–Raphson method

### 1.4.2.6 Dynamic Relaxation

This method is used for large deformation and post-critical. In this method, an artificial time parameter is introduced. Considering inertia and damping, the problem is treated as a dynamic one, using the explicit time integration method. The stiffness matrix is never inverted in this approach. The method also contains Rayleigh damping, which can be set by constants $\alpha, \beta$ according to the formula

$$\boldsymbol{M}\frac{\mathrm{d}^2\boldsymbol{u}}{\mathrm{d}t^2} + \boldsymbol{C}\frac{\mathrm{d}\boldsymbol{u}}{\mathrm{d}t} + \boldsymbol{K}(\boldsymbol{u})\boldsymbol{u} = \boldsymbol{f}, \tag{1.1}$$

$$\boldsymbol{C} = \alpha\boldsymbol{M} + \beta\,\mathrm{diag}[K_{11}(\boldsymbol{u}), K_{22}(\boldsymbol{u}), \ldots, K_{nn}(\boldsymbol{u})], \tag{1.2}$$

where $\boldsymbol{M}$ is the lumped (diagonal) mass matrix, $\boldsymbol{C}$ is the diagonal damping matrix, $\boldsymbol{K}$ is the stiffness matrix, dependent on the solution $\boldsymbol{u}$ in the nonlinear case, $\boldsymbol{f}$ is the vector of external forces, $\boldsymbol{u}$ is the discretized displacement and rotation vector, $n$ is the dimension of $\boldsymbol{u}$.

## 1.5 Dynamic Analysis

Dynamic calculation is available in the add-on module RF-DYNAM. Dynamic analysis contains:

- Eigenvectors
- Forced vibrations
- Equivalent loads

Forced vibrations choice can further include the following loadings

- Time history analysis
- Response spectrum method
- Multi-point response spectrum method

The response spectra method is among others used for seismic analysis. It has two possibilities:

- Linear response spectra method
- Nonlinear (push over) method

Rayleigh damping or Lehr's damping is possible to select. Rayleigh damping was described in the previous paragraph. Lehr's damping (see [1], p.264) allows the user to specify the relative damping $D_i$ of each mode independently according to equation

$$\frac{\mathrm{d}^2 q_i}{\mathrm{d}t^2} + 2D_i\omega_i\frac{\mathrm{d}q_i}{\mathrm{d}t} + \omega_i^2 q_i = f_i, \quad (i = 1,\ldots,n), \tag{1.3}$$

where $\omega_i$ is the i-th eigenfrequency, $q_i$ is the unknown solution component corresponding to the eigenfrequency $\omega_i$, $D_i$ $[-]$ is the relative damping corresponding to the eigenfrequency $\omega_i$, and $f_i$ is the corresponding right-hand side.

Time history analysis is either computed by:

- Modal analysis or
- Direct integration method

The superposition rule is either square root of the sum of the squares (SRSS) or complete quadratic combination (CQC). The direct integration method is based on the implicit Newmark method, which is second order accurate in time and absolutely stable. The Newmark method uses a constant time step approach.

The following types of mass matrix definition are available

- Diagonal (lumped) mass matrix
- Diagonal (lumped) mass matrix with rotational elements (improved version)
- Unit mass matrix

## 1.6 Stability Analysis

The stability calculation is available in the add-on module RF-STABILITY. Two methods for stability calculation are available:

- Linear eigenvalue analysis
- Nonlinear stability analysis (nonlinear calculation, load is increased until structure failure, the critical load factor is determined)

For the linear eigenvalue analysis four methods are available:

- Lanczos method
- Calculation of the roots of the characteristic polynomial
- Subspace iteration method
- Improved Conjugate Gradient (ICG) iteration method

## 1.7 Member Calculation

In RFEM the following types of members can be computed:

- Beam (Bending resisting member that can transmit all internal forces)
- Rigid member (Coupling member with rigid stiffness. It is internally modelled by a high stiffness value, which is set automatically by a couple of orders higher than the surrounding construction, which allows the member to behave rigidly and to be numerically stable at the same time.)
- Rib (Downstand beam considering the effective slab width)
- Truss (Beam with moment releases at both ends)
- Truss (only N) (Member with stiffness EA only)
- Tension (Truss (only N) with failure in case of compression force)
- Compression (Truss (only N) with failure in case of tensile force)
- Buckling (Truss (only N) with failure in case of compression force $> N_{cr}$)
- Cable (in compression it loses its stiffness, has to be calculated with the large deformation option only)
- Cable on pulleys (Member on polyline, can only be shifted in longitudinal direction, absorbing tensile forces only (pulley))
- Definable stiffness (Member with user-defined stiffnesses, without necessity to define cross-section)
- Coupling rigid–rigid (Rigid coupling with bending resisting connections at both ends)
- Coupling rigid–hinge (Rigid coupling with bending resisting connection at member start and hinged connection at member end)
- Coupling hinge–rigid (Rigid coupling with hinged connection at member start and bending resisting connection at member end)
- Coupling hinge–hinge (Rigid coupling with hinged connections at both ends – only axial force is transmitted
- Spring (Member with spring stiffness, incl. nonlinear behavior)
- Dashpot (Parallel combination of viscous dashpot and elastic spring i.e. Kelvin–Voigt model. In static analyses, only the spring part is taken into account.)

A large database of cross-sections is available in RFEM. Hybrid beams, which allow more than one material for the cross-section, are also supported. Each beam can be further equipped by a release (= hinge, in general nonlinear) on each side.

## 1.8 Plate and Shell Calculation

Two types of plate theories, with respect to the transversal shear approximation, can be selected:

- Mindlin theory (linear shear theory, default choice)
- Kirchhoff theory (no transversal shear considered, suitable only for thin plates). The Kirchhoff theory is available only for linear calculations.

The types of surfaces / plates in RFEM:

- *Surfaces with material orthotropy* – plates of constant thickness with orthotropic material defined by six elastic constants
- *Surfaces with geometrical orthotropy* – surface with periodically varying thickness. By the so-called smeared stiffness approach, the 8×8 average stiffness matrix is assembled. The list of all plates of these types, supported in RFEM, follows:
  a. Effective thickness (orthotropic material possibility)
  b. Defined by stiffness matrix (This plate type allows the user to define an arbitrary plate defined by an 8×8 stiffness matrix. This definition allows the definition of unsymmetric compositions also. Initial calculation of the stiffness matrix can be made outsideand imported into RFEM easily from Excel (and exported also)).
  c. Coupling
  d. Unidirectional ribbed plate
  e. Bidirectional ribbed plate
  f. Trapezoidal sheet
  g. Hollow core slab
  h. Grillage
  i. Unidirectional box floor
- *Without tension surface* – plate made of isotropic material. The calculation is nonlinear due to the effect occurring, for example, in masonry walls: under tension the plate loses its in-plane stiffness. Bending is handled linearly.
- *Rigid plate* – plate with rigid stiffness. It is internally modelled by a high stiffness value, which is set automatically by a couple of orders higher than the surrounding construction, which allows the plate to behave rigidly and to be numerically stable at the same time.
- *Laminated plate* – RFEM allows the analysis of laminated plates (= multilayered plates). Any number of layers with material orthotropy is allowed. An unsymmetric composition is supported. The transversal shear calculation is based on the Grashoff formula. Stress/strain dependence throughout the plate thickness can be shown graphically in any point of the plate.
- *Safety glass* – safety (laminated) glass is a multilayered plate consisting of glass and foil layers. The elastic properties of glass and foil differ significantly – by a couple of orders. Due to the significant difference in stiffness, there is a significant difference in the angles of the normal lines (the deformed points are connected to create the normal lines in each layer). These special plates are solved by solid elements to handle the situation accurately.
- *Insulating glass units* – in insulating glass units, glass layers (or safety glass layers) are combined with gas layers. Gas is simulated by solid elements (see the section List of All Elements for

more details). Large-deformation calculation is necessary in this case. The calculation allows the consideration of solar flux radiation and infinite ray reflections in the glass unit.

- *Membranes* – surfaces with no bending stiffness (textile, for example). Isotropic as well as orthotropic membranes can be defined. The calculation is nonlinear due to the effect which occurs in real membranes – under compression the membrane loses its stiffness. Membranes can be calculated only with the large deformations. Let us note that RFEM also has the feature of nonlinear form-finding techniques, which allows finding the unknown shape of a membrane surface that is useful in membrane structure design (circus tent design, for example).

Note, that curved surfaces in RFEM are approximated by planar finite elements. In nodes where planar elements are joined, deformations and rotations are forced to be equal. This approximation converges with a finer mesh to exact values on curved surfaces.

## 1.9 Load

Loads in RFEM can be defined in many different ways:

- *Nodal load* (force load, moment load, imposed deflection load, imposed rotation load)
- *Line Load* (force intesity load, moment intesity load, imposed deflection load, load can vary along the line length)
- *Member Load* (force intesity loading, moment density loading, temperature loading, axial strain loading, axial displacement loading, precamber loading, initial/end prestress loading, displacement loading, rotation loading, pipe loading, rotary motion loading, imperfection loading, loading can vary along the member length)
- *Surface Load* (pressure loading, temperature loading, strain loading, precamber loading, rotary motion loading)
- *Solid Load* (force intesity loading, temperature loading, strain loading, buoyancy loading, rotary motion loading)
- *Free concentrated load* – A free concentrated load acts as a force or moment on any location of the surface.
- *Free Line Loads* – A free line load acts as a uniform or linearly variable force along a freely definable line of a surface.
- *Free rectangular / circular / polygon load* – A free rectangular / circular / polygon load acts as a uniform or linearly variable surface load on a rectangular / circular / polygon, freely definable part of a surface.

## 1.10 Support

RFEM features the following kind of supports:

- Nodal support
- Line support
- Surface support (implementation is based on the subsoil implementation described in the next point)
- Subsoil simulation (using two models – Winkler subsoil and Pasternak subsoil, which allows the definition of shear stiffnesses in the subsoil)

Nodal and line supports can act with respect to any degree of freedom ($u_x$, $u_y$, $u_z$, $\varphi_x$, $\varphi_y$, $\varphi_z$), can work in the rotated coordinate system, and it is possible to add linear springs in any direction.

Nodal supports are further equipped with many nonlinearities like nonlinear springs defined by a multilinear diagram, by partial activity behavior (not acting in the case of positive or negative force in a particular direction), or by friction (stiffness dependence on force in a particular direction). Linear and surface supports are further equipped by partial activity behavior (not acting in case of positive or negative force in a particular direction).

## 1.11 Release

Two types of releases are available in RFEM:

- Nodal release
- Line release

A release (= hinge) between two nodes allows the connection of any degree of freedom ($u_x$, $u_y$, $u_z$, $\varphi_x$, $\varphi_y$, $\varphi_z$), allows the definition of linear stiffness between any degree of freedom, and is also equipped with nonlinearities: nonlinear hinge behavior defined by a multilinear diagram, partial activity behavior and fixed-under-condition behavior (fixed, if $M_y$ is positive, for example). A special possibility is a scissor release. With a scissor release, you can model crossing beams. For example, there are four members connected at one node. Each of the two member pairs transfers moments in its continuous direction, but they do not transfer any moments to the other pair. Only axial and shear forces are transferred in the node. A line release in RFEM allows linear behaviour only.

## 1.12 Material Models

RFEM contains an extensive database of materials from various technical standards and company specifications (steel, concrete, glass, foils, wood). Special materials available in RFEM are gasses (in RFEM solved using solid elements, see the description of finite elements used in RFEM below) and materials with temperature dependent properties. From the modelling point of view, all materials can be modelled using different material models:

- Linear material models:
  - Isotropic Linear Elastic
  - Isotropic Thermal-Elastic (elastic parameters are temperature dependent)
  - Orthotropic 2D (calculations of wood, carbon fiber materials etc.)
  - Orthotropic 3D (calculations of wood, carbon fiber materials etc.)
- Nonlinear material models:
  - Isotropic Plastic 1D (fully plastic model with plastic strains, arbitrary cross-sections)
  - Isotropic Plastic 2D/3D (fully plastic model with plastic strains, the von Mises hypothesis)
  - Nonlinear Elastic 1D (nonlinear calculation without considering plastic strains)
  - Nonlinear Elastic 2D/3D (nonlinear calculation without considering plastic strains, according to the following hypotheses: von Mises, Tresca, Drucker–Prager, Mohr–Coulomb)
  - Orthotropic Plastic 2D (plastic calculations of wood, based on the Tsai–Wu hypothesis)
  - Orthotropic Plastic 3D (plastic calculations of wood, based on the Tsai–Wu hypothesis)
  - Isotropic Masonry 2D (calculation of masonry walls using maximum tensile limit stresses)

In practice it is often necessary to model materials which have different yield limits in tension and in compression. The material models which can handle these effects are the following:

- Nonlinear Elastic 1D
- Nonlinear Elastic 2D/3D (only the Drucker–Prager and Mohr–Coulomb hypotheses)
- Orthotropic Plastic 2D
- Orthotropic Plastic 3D

*Isotropic hardening* is implemented in all of the above mentioned material models. The latter three models (Orthotropic Plastic 2D, Orthotropic Plastic 3D and Isotropic Masonry 2D) allow the definition of *bilinear* hardening, all other nonlinear material models allow the definition of *multilinear* hardening, defined by a diagram). Kinematic hardening is not available.

Another feature is the calculation of *cables* (special type of member) and *membranes* (special type of surface), which have no bending and compression stiffness. The calculation process lowers the stiffness in compression, which makes the calculation nonlinear. Large deflection theory (third order theory) is necessary to be used in that case.

## 1.13 Construction Models

The following construction models can be selected:

| Construction Models | Available Unknowns | Plates are Treated as |
|---|---|---|
| 3D | $u_x, u_y, u_z, \varphi_x, \varphi_y, \varphi_z$ | Shells (bending+membrane loading) |
| 2D - XY | $u_z, \varphi_x, \varphi_y$ | Plate (bending loading only) |
| 2D - XZ | $u_x, u_z, \varphi_y$ | Wall (membrane loading only) |
| 2D - XY | $u_x, u_y, \varphi_z$ | Wall (membrane loading only) |

Table 1.1: Construction Models in RFEM

## 1.14 Appendix - Second Order Analysis

Second-order analysis is on the half way between the first-order theory (small deformation theory) and the large deformation analysis. The small deformation theory is calculated in the undeformed configuration, the resulting equations are linear and therefore no iterations are needed. The large deformation analysis, on the other hand, is calculated in the deformed configuration, which is updated in each iteration. The resulting equations are therefore nonlinear. As part of the large deformation theory, the effect of axial forces on the bending stiffness is considered (the so-called P-delta/P-Delta effect). Tensile forces increase the bending stiffness, compressive forces decrease the bending stiffness. The second-order analysis is calculated in the undeformed configuration, however, the effect of axial forces on the bending stiffness is considered. If axial forces are supposed to be known, then the problem is linear and can be therefore solved within one iteration. If axial forces are unknown, which is the usual case, the problem is solved iteratively as a sequence of linear problems, in which the axial force is updated from the previous iteration and considered constant within the particular iteration step. This numerical approach is in fact the fixed-point iteration method, in RFEM called equivalently the Picard method. The Newton method is not available for this type of analysis[1]. Let us clarify the second-order analysis on the beam equation (see Petersen, *Statik und Stabilität der Baukonstruktionen*, 1982, p.187)

$$\frac{\mathrm{d}^4 w(x)}{\mathrm{d}x^4} + \underbrace{\frac{N}{EI_y}\frac{\mathrm{d}^2 w(x)}{\mathrm{d}x^2}}_{\text{second-order effect}} = \frac{q(x)}{EI_y} \tag{1.4}$$

[1] The Newton method could be used in general. This would require to express the axial force *N* in terms of unknown deflections and rotations, the resulting equations would be nonlinear. However, this approach is not used in RFEM.

where $w(x)$ is the beam transversal deflection, $N$ is the axial force ($N > 0$ means tensional force), $E$ is Young's modulus, $I_y$ is the bending moment of inertia and $q(x)$ is the pressure load. We easily see that by considering the axial force $N$ constant, the equation is linear. If the axial force $N$ is zero, then the equation (1.4) reduces to the standard Euler–Bernoulli beam equation.

In RFEM the second-order theory is available not only for beams, but also for plates and solids.

# 2 Finite Elements

## 2.1 Finite Elements from Topological Point of View

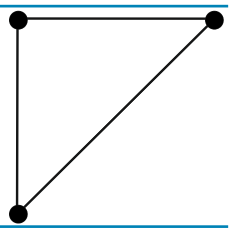
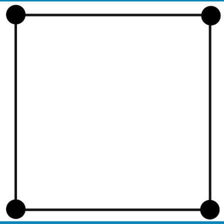
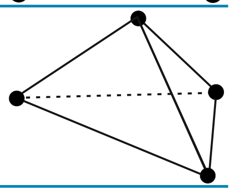
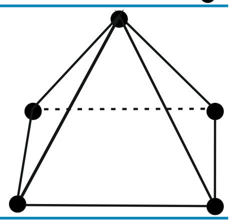
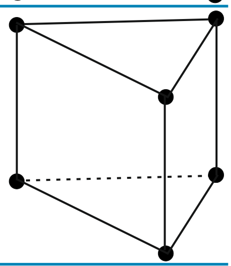
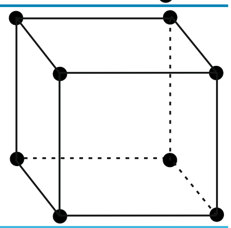
| Dimensions | Name | Number of Nodes | Sketch |
|---|---|---|---|
| 1D | beam | 2 |  |
| 2D | triangle | 3 |  |
| | quadrangle | 4 |  |
| 3D | tetrahedron | 4 |  |
| | pentahedron – pyramid | 5 |  |
| | pentahedron – prism | 6 |  |
| | hexahedron – brick | 8 |  |

Table 2.1: Construction Models in RFEM

## 2.2 List of All Elements

The finite element types used in RFEM are given in the following table. They are chosen automatically by the program according to the situation.

| Dimensions | Element Type | Element description |
|---|---|---|
| 1D | beam element | element with rotational degrees of freedom |
| 2D | plate element | Lynn–Dhillon |
| | | MITC3 |
| | | MITC4 – used in case of nonlinear calculation |
| | wall element | with stabilized zero-energy modes |
| | shell element | shell element = plate element + wall element |
| 3D | solid element | element with rotational degrees of freedom |
| | | element without rotational degrees of freedom (with or without extra shape functions) |
| | | gas element |
| | | contact element |

Table 2.2: Element description

## 2.3 Integration Procedure

For members, analytical integration is used for linear cases, while in the nonlinear setting the two point Gauss quadrature is used along the beam. For nonlinear cases the following integration rule is used in the cross-section: 2×2 Gauss quadrature for quadrangles and 4-point selective reduced integration rule for triangles (3 points for $\epsilon_x$, $\epsilon_y$ and 1 point for $\gamma_{xy}$).

In plate elements, analytical integration is used whenever possible (in Lynn–Dhillon element or in a triangular element). In other cases, a 2×2 composite Gauss quadrature is used in the element plane (quadrangles). In solids, a 2×2×2 composite Gauss quadrature is used in hexahedrons. Reduced one point integration is used for some particular terms to avoid numerical problems.

Let us focus on integration in plates with respect to their thickness, which is based on the Gauss–Lobatto quadrature. The Gauss–Lobatto quadrature is a Gauss quadrature in which boundary points are forced to also be integration points, which allows an exact evaluation of stresses on layer interfaces in case of multilayered plates. In case of linear calculation, three integration points are used per layer. In nonlinear calculation, nine integration points are used in the plate (nonlinear calculation allows one layer only).

## 2.4 Mesh Settings

Two mesh options exist in general:

- mapped meshing (= structural meshes)
- unmapped meshing

In the mesh settings, the user can define the element size, preference of the mapped meshing, and additionally has the possibility to generate only triangles or only quadrangles in surface meshes. Mapped meshes should be preferred due to better accuracy of stress results. In solid analysis for example, tetrahedrons give less accurate stress results comparing to analysis made on hexahedrons.

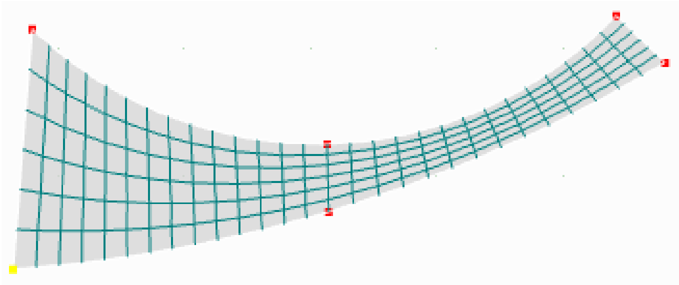


Figure 2.1: Mapped finite element mesh

## 2.5 Preventing Zero-Energy Modes

The following elements in RFEM can theoretically exhibit zero-energy modes: 3-node and 4-node quadratic wall elements, and all solid (4-, 5-, 6-, 8-) node elements with rotational degrees of freedom. In RFEM the so-called penalty stiffness is added. The stiffness is so small as not to influence results, but sufficiently high to prevent zero-energy modes.

## 2.6 Conforming Versus Non-conforming Elements

Non-conforming elements are those whose deformations and rotations are not continuous between elements:

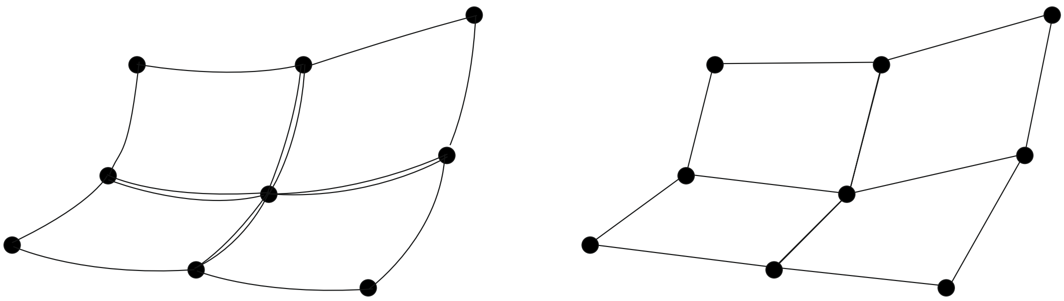


Figure 2.2: Left: non-conforming finite element, right: conforming finite element

The only non-conforming elements in RFEM are solid elements, which contain extra shape functions (ESF), i.e., linear solid elements without rotational degrees of freedom and linear wall elements.

## 2.7 Beam Finite Element

Beam (= member) finite elements contain displacement as well as rotational degrees of freedom.

## 2.8 Surface Finite Element

For bending components (i.e. for model plate and for bending components in a shell model) the following three finite element types are used:

- Chirkov element (Kirchhoff plates, linear or Picard calculation)
- Lynn–Dhillon element (Reissner–Mindlin plates, linear or Picard calculation, used up to RFEM 12.02.138917)
- MITC3 element (replacing the Lynn–Dhillon element from RFEM 13.01.140108)
- MITC4 element (Kirchhoff and Reissner–Mindlin plates, the Newton–Raphson calculation)

For membrane components (i.e. for model wall and for membrane components in model shell) the wall element is used. In shells, both elements for bending and membrane components are combined.

### 2.8.1 Chirkov Plate Element

The Chirkov element is a planar four-node element composed of four triangle elements used in case of Kirchhoff plates and linear calculation or nonlinear calculation based on the Picard method. The mid-element node is eliminated. The element uses the mixed formulation. Its unknowns are three displacements and three rotations in each node. For linear calculations, the Chirkov element is more precise than the MITC4 element. However, the eliminated node, which cannot be loaded by any force (it is eliminated under assumption of no loading), brings instability when it is used together with the geometric or material nonlinearity, where the Newton–Raphson method or other nonlinear iterative method is used. In this case the MITC4 element is used.

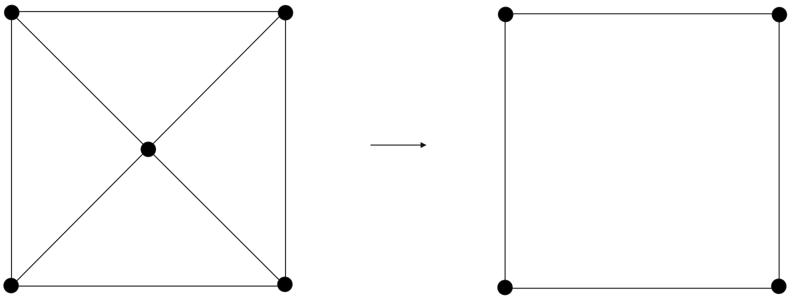


Figure 2.3: Middle node elimination.

For this element, the analytical integration method is used, therefore no zero-energy modes exist. Shear locking cannot occur in Kirchhoff plates.

### 2.8.2 Lynn–Dhillon Plate Element

The Lynn–Dhillon element is also a planar four-node element, composed of four triangle elements used in case of Mindlin plates and linear calculation or nonlinear calculation based on the Picard method. The mid-element node is eliminated. For linear calculations the Lynn–Dhillon element is more precise than the MITC4 element. However, the eliminated node, which cannot be loaded by any force (it is eliminated under assumption of no loading), brings instability when it is used together with geometric or material nonlinearity, where the Newton–Raphson method or other nonlinear iterative method is used. In this case the MITC4 element is used.

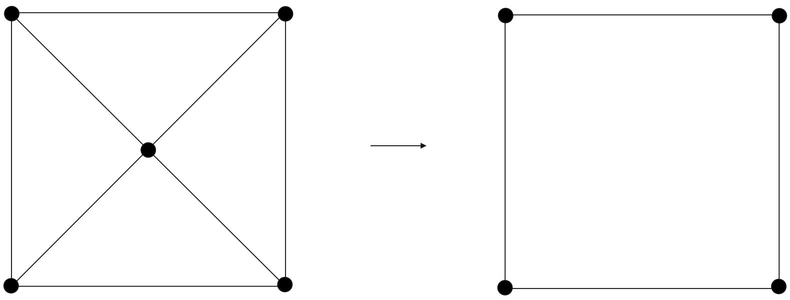


Figure 2.4: Middle node elimination.

For this element, the analytical integration method is used. Therefore no zero energy modes exist. Shear locking can occur in case of very thin plates, for this case the bound for shear stiffness is applied to avoid this problem.

### 2.8.3 MITC3 Plate Element

The MITC3 is an isotropic triangular element exploiting the mixed interpolation of tensorial components for the transverse shear terms to avoid shear locking, hence it is a more robust finite element, than the Lynn-Dhillon one. The price to pay is the lower order of approximation (linear, instead of quadratic), therefore a finer mesh is needed to achieve the same precision of results. It exploits the full 3-point Gauss integration for the stiffness matrix on the elements. In case of quadrangles, the aforementioned middle-node elimination technique is used for four triangles.

Since RFEM 13.01.140108, MITC3 fully replaces the Lynn–Dhillon element, namely in linear static analysis, linear stability analysis, linear dynamic analysis, and nonlinear static analysis in case of first and second order theory without material nonlinearity and foundation.

### 2.8.4 MITC4 Plate Element

The MITC4 element is an isoparametric planar four-node element. A less precise element compared to the Lynn–Dhyllon element, but more robust in case of nonlinearities. The element uses linear base functions. For this element the full Gauss quadrature is used (2×2 integration points), which yields no zero-energy modes. Elimination of shear locking at decreasing thickness is done by the mixed interpolation of deflection, rotation and slope.

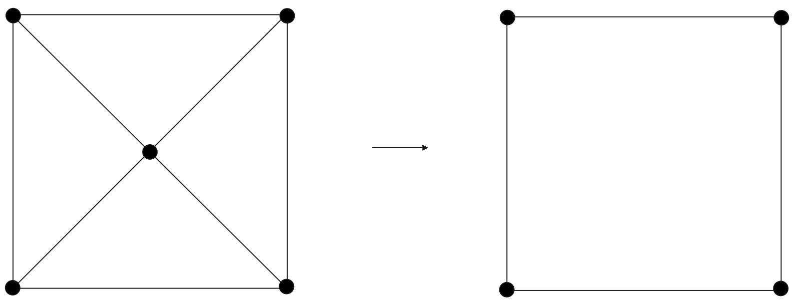


Figure 2.5: MITC4 element.

### 2.8.5 Wall Element

The element contains two displacements and one rotation per vertex. For walls and shells, the quadratic polynomial element with rotational degrees of freedom is used. This element is derived from eight-node/ six-node quadratic isoparametric element by mid-side nodes elimination. This element can theoretically suffer from zero-energy modes. In RFEM the so-called penalty stiffness is added. The stiffness is small enough not to influence results, but sufficiently big to prevent zero-energy modes.

We can call this element quadratic, because it is based and derived from the genuine quadratic element, however, some additional simplifications are introduced so that the quadratic polynomial within the element is reduced. As a result, this element is more precise than linear isoparametric bricks, but less precise than genuine quadratic isoparametric bricks. On the other hand, computationally it is much more efficient contrary to the genuine quadratic element, because it contains no mid-nodes. If rotational degrees of freedom are not considered, a linear element with extra shape functions is used. The extra shape functions are used to avoid shear locking effects.

## 2.9 Solid Finite Element

All elements have three displacements and three rotations defined in each node, i.e., six unknowns per node. If setting *Ignore rotational degrees of freedom* in the *Global Calculation Parameters* dialog, then rotational degrees of freedom are not supported, all elements have only three unknowns per node. From all the elements listed above, hexahedrons are the most accurate and should therefore be preferred, if the construction allows the use of a mapped mesh. Especially tetrahedrons can offer less accurate results for stresses.

All solid elements which have both displacement and rotational degrees of freedom, can theoretically suffer from zero-energy modes. In RFEM, the so-called penalty stiffness is added. The stiffness is small enough not to influence results, but sufficiently big to prevent zero-energy modes.

Let us describe in more detail the hexahedron / brick elements. In RFEM the following two types are used (note that 4-, 5- and 6-node elements are analogously divided into the same two categories):

- hybrid brick elements with rotational degrees of freedom, used for the shell model
- linear isoparametric brick elements, used for the membrane model (with or without extra shape functions, also called bubble functions)

### 2.9.1 Quadratic Solid Element with Rotational Degrees of Freedom

Let us describe this kind of element on hexahedron. Hexahedrons have eight nodes, in each of which six independent components are defined (three displacements and three rotations), hence the element has altogether 48 unknowns. It is derived from the isoparametric quadratic element, which has nodes not only in vertices but also in the mid-sides. We can call this element quadratic, because it is based and derived from the genuine quadratic element, however some additional simplifications are introduced, so the quadratic polynomial within the element is reduced. As a result, this element is more precise then linear isoparametric bricks, but less precise then genuine quadratic isoparametric bricks. On the other hand, computationally it is much more efficient contrary to the genuine quadratic element, because it contains no mid-nodes.

### 2.9.2 Linear Solid Element with or Without Extra Shape Functions

Let us focus on hexahedrons. Hexahedrons have eight nodes, in each of which three unknown displacements are defined (no rotational degrees of freedom). The base functions are linear. From a theoretical point of view, it is a linear isoparametric element, which allows extra shape functions (ESF, also called bubble functions). Extra shape functions are extra quadratic base functions, the corresponding unknowns of which are eliminated from the element, so they do not add any new unknowns. Their benefit is to increase the accuracy of elements in bending.

### 2.9.3 Solid Gas Element

The solid gas element is used for gas calculation. Its formulation is based on the ideal gas law, which under constant temperature takes the form $pV = \text{const}$. This element can be used only under large deflection analysis.

### 2.9.4 Solid Contact Element

Solid elements allow the modeling of contact behavior between two surfaces. The possible contact behaviors in RFEM are as follows:

| Contact Element Setting | Schematic Diagram |
|---|---|

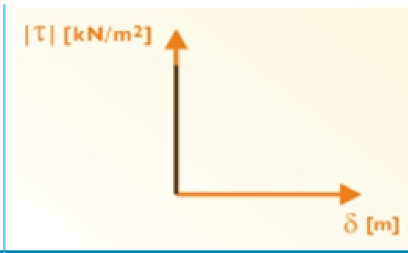
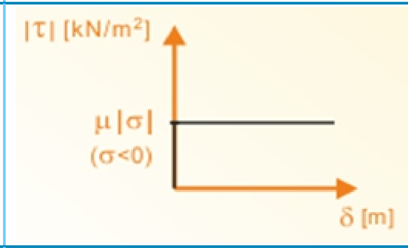
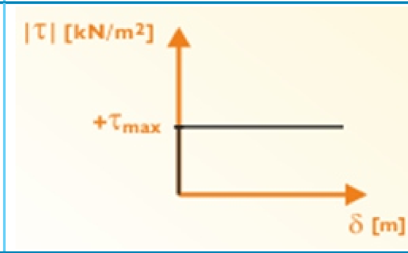
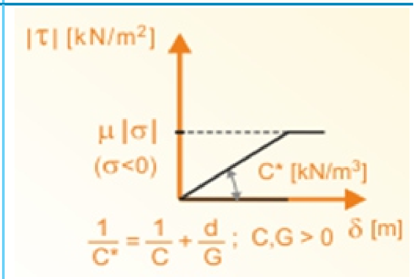
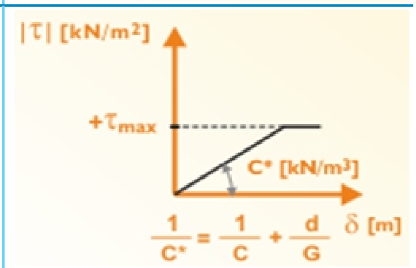
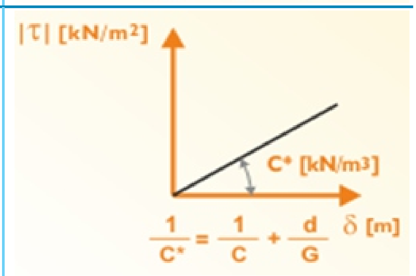
<table>
<tr><td>Full force transmission (no contact active)</td><td>
</td></tr>
<tr><td>Behavior based on the Coulomb friction law</td><td>
</td></tr>
<tr><td>Behavior based on the maximum shear stress definition</td><td>
</td></tr>
<tr><td>Behavior based on the Coulomb friction law (with numerical elastic stabilization)</td><td>
</td></tr>
<tr><td>Behavior based on the maximum shear stress definition (with numerical elastic stabilization)</td><td>
</td></tr>
<tr><td>Elastic solid behavior</td><td>
</td></tr>
</table>

Table 2.3: Solid Contact Element

# Literature

[1] Zdeněk Bittnar and Jiří Šejnoha. *Numerické metody mechaniky*. Vydavatelství ČVUT, Zikova 4, Praha 6, 1992.

[2] Ivan Němec and Vladimír Kolář. *Finite Element Analysis of Structures - Principles and Praxis*. Shaker Verlag, Aachen, 2010.

# Index