Chapter 6
Finite-Element Simulation

The Finite-Element formulation of Musical instruments has the advantage, that the definition of elements is very sophisticated. So we may define elements to be very small or to cover small sections of the geometry which is more difficult in the Finite-Difference formulation when not applying virtual points within the structure.

Finite-Elements were historically constructed for static applications. The formulation of a linear equation system with a stiffness matrix, a vector of the dependent variables we want to solve for, and the forces acting from outside then made it easy to also formulate an eigenvalue formulation of the system. The time-dependent methods of Finite-Elements which are common today also are very elaborated, but only implicitly solve the stiffness matrix which is very expensive. Another problem with time-dependent FEM solutions are the solvers used, which are built for problems of fast conversion. If e.g. within a room a working machine is getting hot and an air-conditioning is cooling it down, one will be interested in the temperature the system will end up. So here, time-dependent solvers try to get to a converged solution using as few time steps as possible. In Musical Acoustics on the other hand we are not interested in any conversion but in the precise development of the waves. As the time-stepping algorithms of FEM used today mostly try to achieve conversion by using strong damping of the systems, normally high frequencies are not reliable with FEM simulations and explicit FDM methods are the best choice here.

With the use of commercial FEM applications the use of FEM has changed, too, compared to the old style using punched cards, where only a few elements were possible to calculate. There, the art of engineering was to use and also to invent only a few elements which fit the shape to analyze most effectively. Large areas with only small displacements may therefore have only one or two finite elements, where sections of strong displacements may have more. Also it is possible to use different element types within one problem, quadratic, round, cylindrical, even arbitrary shapes if necessary and analytically possible. This leads to very few node points over the structure which may indeed solve the problem very intelligent and efficiently. Today, using commercial software with automatic mesh generators, often hundreds of thousands of elements are used and can be generated with a few mouse clicks. Still this
leads to very big stiffness matrices, where the converged solution of the equation system may contain errors at sharp edges. This comes from the fact that although at such edges the error may be considerable, the large amount of elements used at geometrical areas which are smooth in geometry, and therefore the solution nearly perfectly fits there, balance the overall error. So meshing and the use of elements need to be considered carefully with FEM applications, and using more elements may not lead to better results.

6.1 Method

We want to show the method of Finite Elements by looking at a plain 2D stress-strain model of a rectangle element. The differential equation of such a problem is

$$ D C \, D_\varepsilon, u = -p $$

(6.183)

This is Newton’s second and third law of action-reaction of forces and of these forces having opposite sign. So the left hand side are the forces within the body because of its strains caused by its displacements, and on the right hand side \(-p\) is the vector of the pressure as force per area, acting on the body from outside. \(u\) is the vector of the displacements which are the dependent variables we want to solve for. \(C\) is the material matrix containing the material data, like Young’s modulus \(E_x\) and \(E_y\) in x- and y-direction and the Poisson contraction number \(\nu \sim 0.3\). In a 2D application it is

$$ C = \frac{1}{1 - \nu^2} \begin{bmatrix} E_x & \nu E_y & 0 \\ E_y & \nu E_x & 0 \\ 0 & 0 & E_{xy} \frac{1-\nu}{2} \end{bmatrix} $$

(6.184)

like all other matrixes covering the contraction, the Poisson contraction, and the shearing. The differential matrices are

$$ D = \begin{bmatrix} \frac{\partial}{\partial x} & 0 & \frac{\partial}{\partial x} \\ 0 & \frac{\partial}{\partial y} & \frac{\partial}{\partial y} \end{bmatrix} $$

(6.185)

and

$$ D_\varepsilon = \begin{bmatrix} \frac{\partial}{\partial x} & 0 & \frac{\partial}{\partial x} \\ 0 & \frac{\partial}{\partial y} & \frac{\partial}{\partial y} \end{bmatrix} $$

(6.186)

representing the differentials.

So now we have the governing differential equations using \(u = \{u, v\}\) with \(u\) and \(v\) as the displacements in x- and y-direction. First they are differentiated with respect to x and y according to \(D_\varepsilon\) to get the strain \(\varepsilon\) in both directions and in the other direction respectively. So then we have the stress which is multiplied with the according material constants in the \(C\) matrix. All those equal the pressure values on the rhs. The last step is applying \(D\), which