and the stiffness of the structure is k. Having to deal in this case with only one degree of freedom, if u = 1 (unitary displacement) we obtain that k = F, i.e. k represents the force with which the spring reacts to the imposition of a unitary displacement.

We then recall that [K] is a symmetrical matrix and therefore $K_{ij} = K_{ji}$.

It should also be noted that if the structure has n degrees of freedom, the global stiffness matrix size is n x n.

7.2.1 Superelements

The Finite Element Method has theoretical origins that can be traced back to the early '900, but it was realized that, in order to have a practical use, it was necessary to have an instrument capable of solving systems of equations of considerable size. The first calculators, however, had rather low capacities, even in terms of memory. Therefore the number of equations, and therefore degrees of freedom, that constituted the system to be solved could not be very high.

At that time an artifice was frequently used to obtain reliable results in the areas of interest, where the mesh must be adequately dense (see Chapter 6), keeping the overall size of [K] below the computer's capabilities.

The expedient consists in dividing the complete structure under analysis into subsets; for each of these blocks a finite element model is created which has the nodes lying on the separation lines (or surfaces) coincident with those of the adjacent subset; for each block a "reduced stiffness matrix" is then determined at the interface nodes which represents the elastic behavior, at the points where it is calculated, of the substructure to which it refers (this method is also known as the "superelement technique").

Let's make an example: we have a computer capable of solving at most a system of order 12000 (12000 degrees of freedom); this means that, if we use elements whose nodes have 6 degrees of freedom each (such as shell elements and beam elements), we can use at most 2000 nodes to model the structure to be analyzed; Then suppose that a satisfactory mesh leads to 3000 nodes without any further possibility of reduction (not to penalize too much the accuracy of the results), but can be easily divided into two parts, each consisting of approximately 1500 nodes, along a line on which lie for example 20 nodes. With these assumptions we are not able to solve the entire problem, but we can handle separately the two groups, provided we take into account the fact that the two superelements are connected through those 20 nodes.

Nowadays, as mentioned above, thanks to the increasing availability of more and more powerful and cheap computers, this problem is much less felt, since it is possible to easily solve even structures with a few million degrees of freedom.

However the superelement technique can be profitably used, for example, when two different companies (perhaps using two different calculation codes) are analyzing two parts of the same structure which are interfaced: to verify separately and correctly the two subsets it is sufficient to exchange the respective stiffness matrices reduced to the interface points. In the light of this important aspect, in the following we will illustrate how it is possible to obtain such a matrix, if the calculation code being used is not able to carry out the necessary operations automatically, and what the differences are in the case in which it is decided to neglect one of the two parts of the structure, binding the group of interest in the interface points, for example with clamps (an operation, this, which would lead to having theoretically infinite stiffnesses in such nodes).

7.2.2 A practical example

The structure we propose to analyze is the support shown in figure 7.1; it is connected along the vertical plate to a sheet metal by three bolts; the sheet metal in turn is welded along its vertical sides to a very rigid structure. The bracket is loaded at the two holes located in the horizontal plate with two forces acting along the vertical direction, oriented upwards and having a modulus equal to 30 kN. The bracket is made of brick elements, the plate has been modeled with shell elements, given the small value of the thickness compared to the other dimensions, while the screws have been schematized with rigid MPC type elements. Then, since, as we have said, the sheet metal is welded to a very rigid structure, we can impose clamp constraints on the nodes lying along its vertical contours (see figure 7.1).

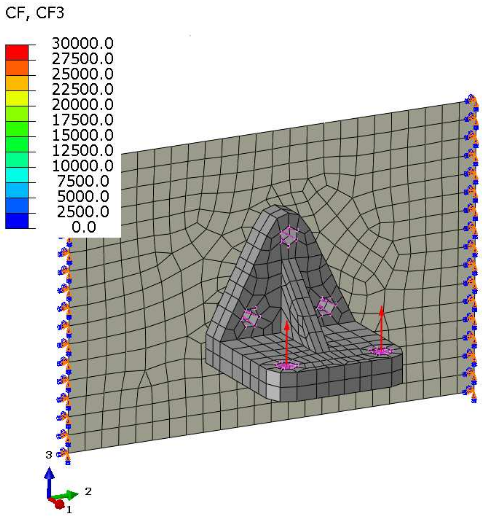

Figure 7.1. Finite element model of a support. We observe the constraints along the vertical edges of the plate and spiders of MPC elements at the holes.

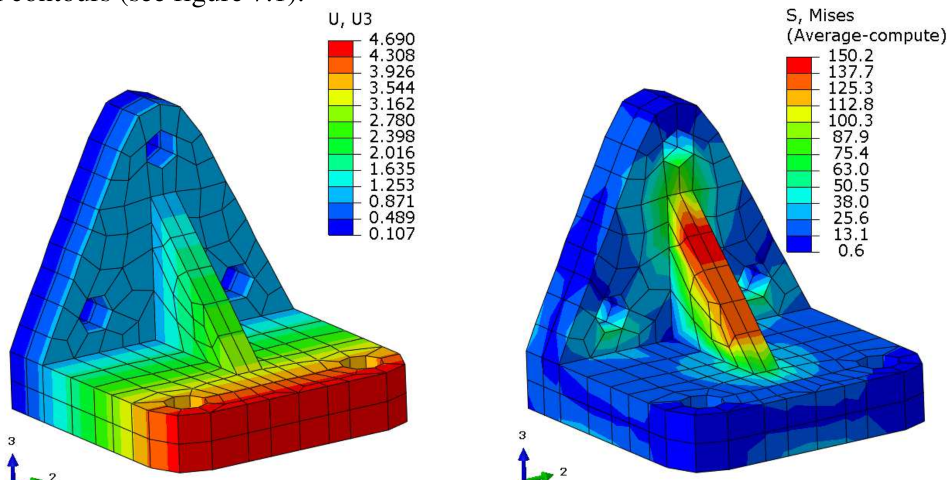

Figure 7.2. Vertical displacement (left) and equivalent Von Mises stress (right) for the support (sheet metal not shown).