NON-LINEAR PROBLEMS OF STEEL CIVIL ENGINEERING STRUCTURES

THE NON-LINEAR PROBLEMS OF THE STEEL CIVIL ENGINEERING STRUCTURES SOLVED BY ANSYS PROGRAM

Milan Vasek

Dpt. Of Steel Structures, Faculty Of Civil Engineering, Czech Technical University, Prague, Czech Republic

---

ABSTRACT

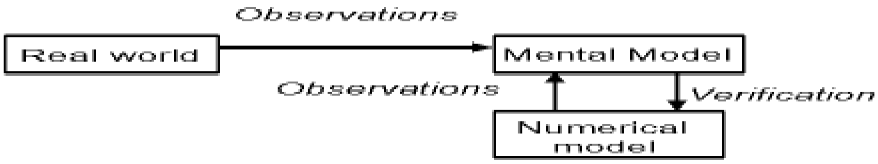
The civil engineering structures are today designed with respect to the limit state of serviceability and limit states of the strength and stability. The design of more efficient and light structures leads to the nonlinear analysis of deflections and stresses. Some parts of structure could reach the yield point under the design load. Some parts of structure could lose their local stability. These complex problems of a different nature are possible to solve by FEM methods. The material constants of the model and values of constants for contact elements, are the crucial points of such a calculation. The structural systems, which are mainly used in civil engineering, are plane and/or space frameworks. Any solution of thin-walled structural systems has to deal with the imperfect products. All these imperfections highly affect deflections and stresses at the nonlinear calculation. The structural parts, which strongly influence the behavior of frames and trusses, are joints of members. These parts are usually semirigid. We obviously apply either rigid or pinned connection for the analytical model. The result of such calculations gives values that are in most cases safe but not economical. The stability solution of such structures, especially for space frameworks is usually not that clear. The stability behavior is affected by the joint rigidity. The FEM calculation of joints help to establish their rigidity, however we still have to make some laboratory testing of joints to calibrate FEM model. In some types of connections the contact between surfaces is changed within the loading process and the distribution of deformations and stresses is highly dependent on the area of contact and acting force. Some experiments are necessary to provide to calibrate the computer solution. The serviceability state of the structure depends on the deflection of joints. The deflection of joints is usually composed from the elastic and plastic part. Any definition of limit state should take into the account the deformation energy spent over the deflected region of a structural joint. The complex nonlinear solution of the 3D beam systems is introduced. The effects of different terms of the energy are clearly separated and the influence of joint's rigidity can be applied too. The following FEM studies with ANSYS program were done by the author within last 4 years under the research activities at the Department of Steel Structures of the Faculty of Civil Engineering, Czech Technical University.

THE WELDED BRACKET FOR "Z" PURLINS

The description of the problem

The purlins of roof structure of industrial halls and of some other similar type of buildings are often designed from light profiled beams cold formed from thin metal sheet. The thickness of these girders is in the range from 0.75mm to 3mm usually. The local stability of flanges and web has to be checked and also the lateral buckling together with the local stability has important effect on the carrying capacity of the purlin. The influence of the reaction at the support of the purlin is very important with respect to the slenderness of the web. The supporting bracket is usually casted. The price of whole casting process and the form is not cheap. The possibility to use welded bracket was studied and some results are described in the following. Two bolts hold the bracket. The contact is developed between the lower flange and top chord of the roof beam. Four bolts hold the Z purlin on the vertical flange of the bracket. The stiffener between flanges is welded.

Figure 1. The FEM model of the bracket with an example of loads and reaction distribution



The FEM model of the welded bracket

The metal sheets used for the bracket are 6mm and 8mm thick. The SHELL43 element was used for the FEM model. The distribution of the deflection and the distribution of stresses depend on the contact between the lower plate and the support. The support is an upper flange of the roof beam. The bracket is held by two bolts to the flange of the beam, which is supposed to be rigid enough. The loads are applied at the circular edge of four bolts that are supporting the Z purlins. The distribution of forces and reactions over the bottom flange bolts (as hinge) is shown at the figure 1. The distribution of the reaction on the support was introduced by CONTAC52 element applied at nodes at the bottom flange. The material constant of the CONTAC52 element was introduced as the calculated reaction of the purlin distributed over the bottom flange of the bracket. Some samples of the bracket subjected to the vertical and the horizontal reactions were tested. The vertical reaction was introduced as the tension and the compression with respect to the bottom flange.

The results of the solution

The results correspond well to the results of the experimental samples loaded by the vertical tension and/or horizontal force. The results of FEM calculation of the compressed bracket were necessary to calibrate with the experiments according to the measured deflection of the bracket. To reach relatively good correspondence between the calculation and the experimental results we have to introduce the mesh of high density at the bottom flange.

The initial value of the stiffness constant of the CONTAC52 element was necessary to change also. This allowed having not so rapid change of calculated reactions at the CONTAC52 elements, which lead to the calculation failure.

Experience from the FEM study

The distribution of deflections and stresses depends on the density of contact elements and on the value of the material constant of the CONTAC52 element. The increasing gap between contact surfaces leads to the higher pressure at lasting contacts. The solution became unstable when the refinement of the contact elements (corresponding to the number of nodes of elements at the bottom flange) was small. This mesh refinement corresponds also to the value of the stiffness constant of the contact element. To have more exact conclusion, a more detailed study should be done.

THE CONICAL JOINT FOR A SPACE FRAME MADE BY THE PRESSURE MOLDING MACHINE

The description of the problem

The research on space steel roof was conducted in 1994-1996. Under this research (Vasek, M. Drdacky M., Hoblik, K, 1996) the space coldformed joint was developed. The basic body of joint was made from the hotrolled tube. The cross sections of rolled tubes have not acceptable tolerances from the circle. The part of tube allows connecting several tubular members by bolts in the theoretical point. Such a space joint allows forming grid only.

To form double curvature surfaces we can use also the part of the tube, but we need to change the cylinder to the cone shape. The molding pressure machine can easily do this. The conical pin helps to calibrate cross section. The more accurate shape is necessary for the successful erection of space dome or barrel vault. The first simple conical shape was tested and calculated. The joint was tested at the laboratory. The numerical analysis was done by the ANSYS program. The calculation was used simultaneously with experiments. The necessary stiffeners of the joint were added and the calculation was introduced at first to check the reasonability of the design. The dimensions of the experimentally tested sample were corrected and the experimental test was done after that.

The model of the joint

The SHELL43 element was used for all version of the space joint. The bilinear stress strain relation was used for the steel. The material that was used is ordinary S235 steel. The increase of the yield point due to the cold forming was not considered, because of very small deformation introduced into the joint. The inclination of the cone was about 70 to maximum 90. This angle corresponds to the ductility of the steel. The tube that was used had about 150mm diameter and thickness was 20mm. The stiffeners that were added were from 10mm thick metal sheet. The elongation of the edge of the tube doesn't lead to the rupture of the steel. The forces acting on the body of the joint are distributed along the circular holes corresponding to the bolts that connect the tubular members of the framework. We designed the joint for the six members connected at one joint. The supports were distributed as external loads also at the edge of bolt holes. Several loading cases were solved. One group of loading cases was composed from six forces acting at three holes. We applied compression forces and in the other case tension forces. The configuration was always keeping one hole supported and the next loaded, all around the body of the joint. We also calculate the case with only one force acting as the tension or as the compression to the joint. Each solution was divided to 15 substeps. The large deflection option was included and the stress stiffening also.

Figure 2. The equivalent stresses due to one load applied on the joint

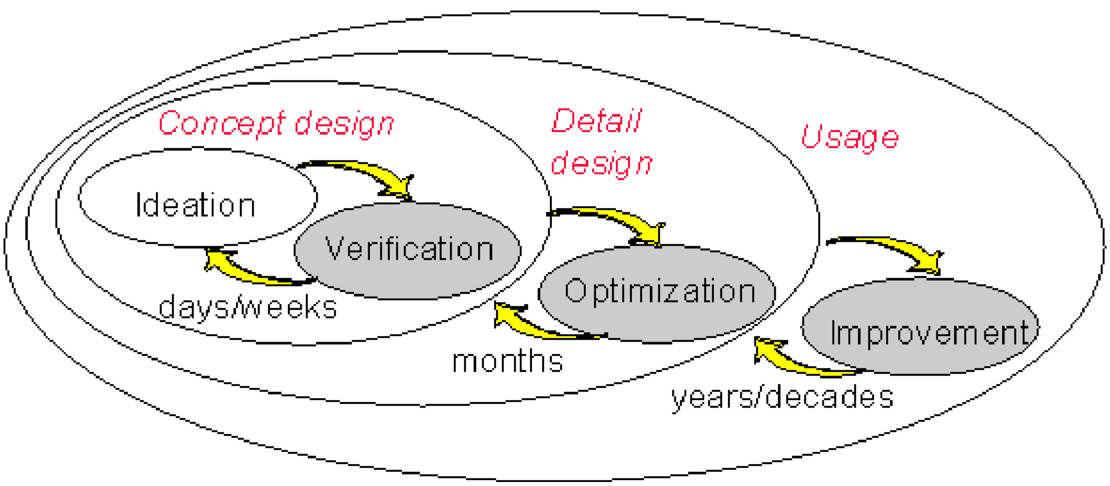
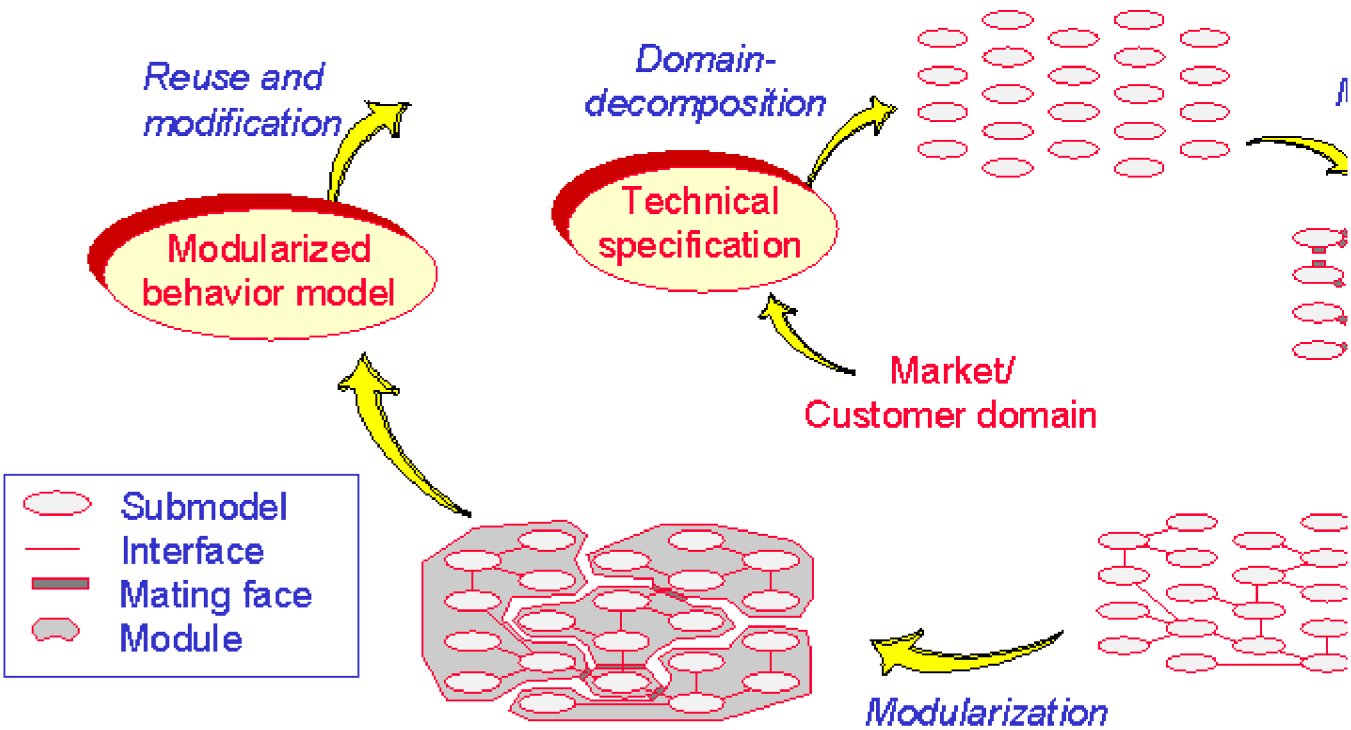


Figure 3. The model and stresses at the cone joint



The FEM solution

The models were prepared mainly on a PC machine. The calculations were done at the School of Engineering at Pittsburgh University, where the author was visiting 1995-1996. The calculation was performed on both Sun stations and on a PC. The calculation of each joint took about 48 hours within the university net. The applied loads were corresponding to the values of loads applied at the experimental tests. The maximum value of the load corresponds to the observed yielding of some regions of joints.

The results of the research

The final result from the study is the space steel joint that was able to carry forces at the general spherical or cylindrical dome up to 300kN. Deflections developed by these forces were admissible and plastic zones appeared only in the vicinity of holes where the loads were acting. The experiments that were done were compared with FEM results. The correspondence was very good in the elastic regions that were measured by straingages. There is no definition available of the local deflection acceptable for the real application.

THE BEHAVIOR OF ONE-SIDE WELDED I-BEAMS WITH IMPERFECTIONS OF WEBS

The description of the problem

The competition between the producers of steel structures and the effort to save material and labor cost to minimum, leads to the usage of some rather not obvious structural details. The one side fillet welds of the I welded section is one of these. The one side fillet welds are possible to use according to the EC 1993-1. The most important condition is that the loads applied to the flanges of the beams are acting without the eccentricity. The tolerances of the geometry of the beam are assumed to be in the range prescribed by codes. Some producers are complaining that the value of tolerances given by codes is based on esthetic. The following study showed that the tolerances are fully justified.

The model of the beam

The real welded beam might has undulation of webs about twice bigger waves then ENV recommends. A large amount of fillet welds could be thinner then declared. To find that these imperfections are acceptable in the case that beam is designed up to 99% of limit bending moment, the FEM model was made. To model the undulation the Microstation 95 program was used. The "bowls" located on the web were arbitrarily modeled as a groups of facets on the surface made from elevated lines forming the rectangles with round corners. The part of the whole frame was constrained to be close to the whole frame. The one side weld was modeled as 450 inclined plane of the weld size, approximately about 4mm. The weld was divided to smaller parts. The wire mesh was transferred through IGES to the ANSYS environment. The areas were created by the ANSYS preprocessor. The areas approximate the continuous undulation as facets with good accuracy. The SHELL43 was used for the solution. The large deflection effect was included and the bilinear curve of the stress strain relation for the obviously used steel S235 was used. The three models were solved:' the one-side welded beam centrally loaded - the one-side welded beam eccentrically loaded - the beam with ordinary connected web and flanges centrally loaded The beam model actually corresponds to the part of gable frame that is the cross section of an assumed industrial hall. Model was loaded by local forces (corresponding to purlins in reality) and by the stresses calculated from the engineer's design. The applied stresses were distributed over the nodes at the ends of the beam.

Figure 4. The stresses at the model of joint with the bottom stiffener and the stiffening collar.

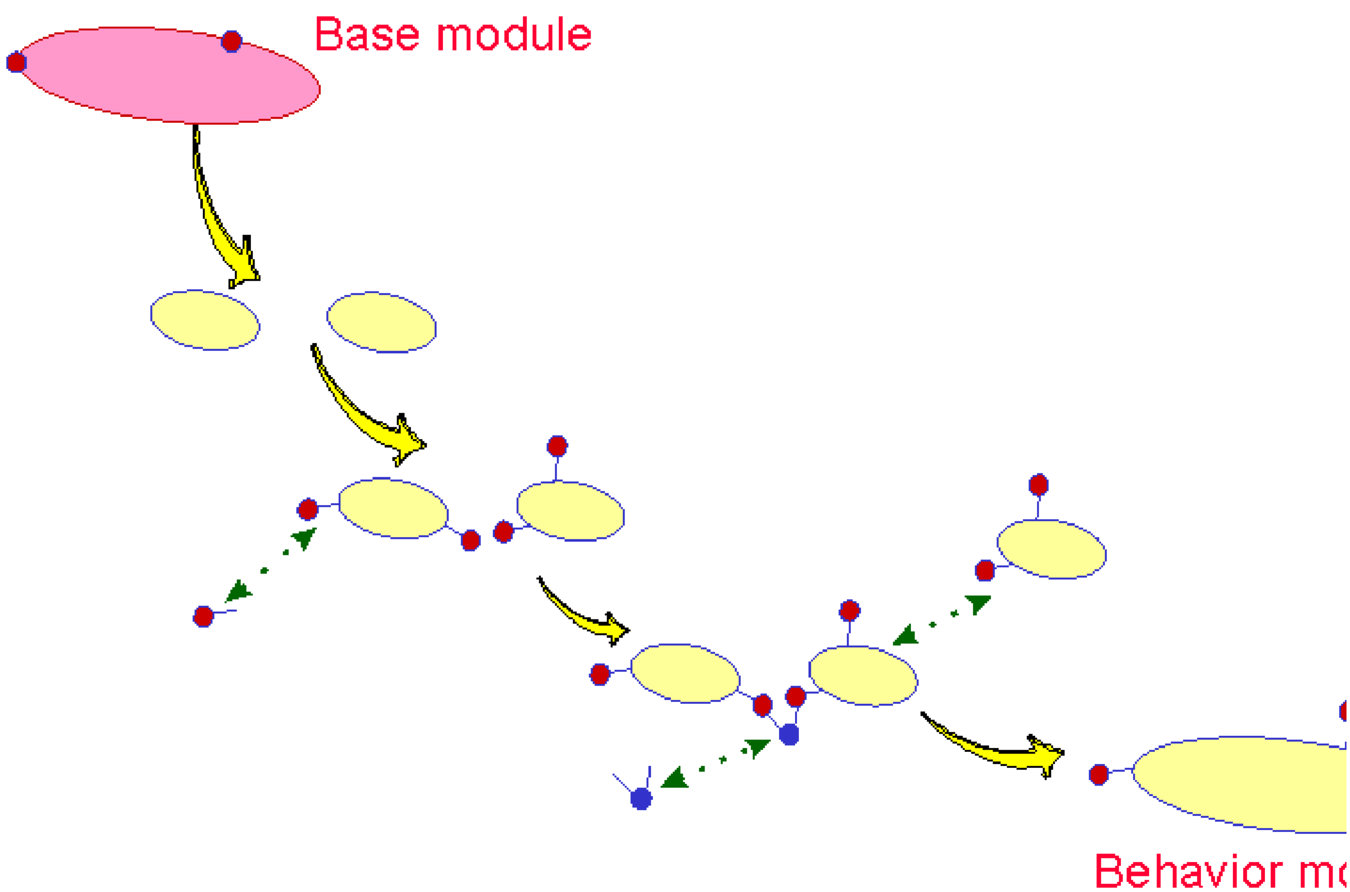


Figure 5. The FEM model of undulated beam

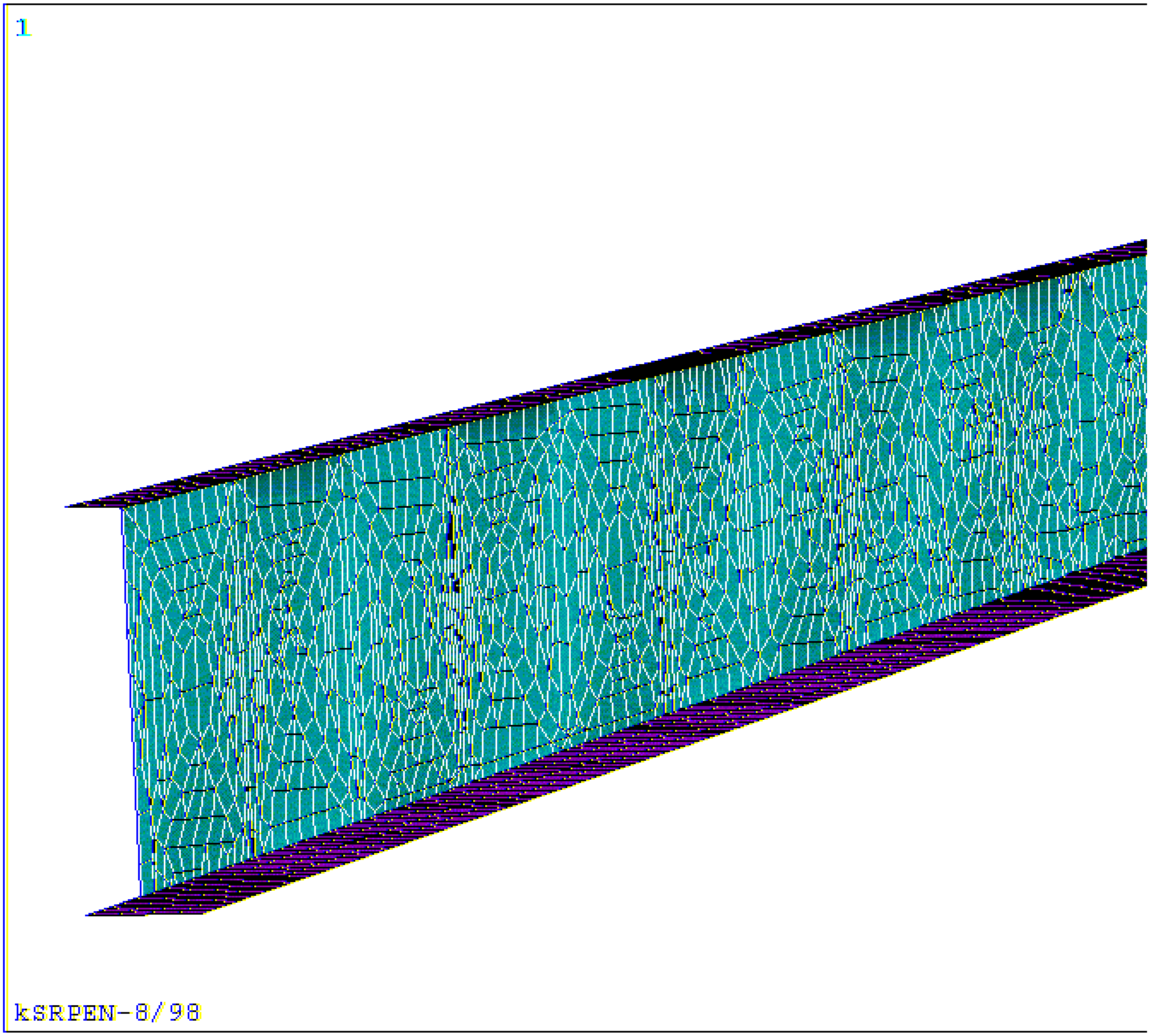


Figure 6. Cross-section of the model of the beam with undulated web

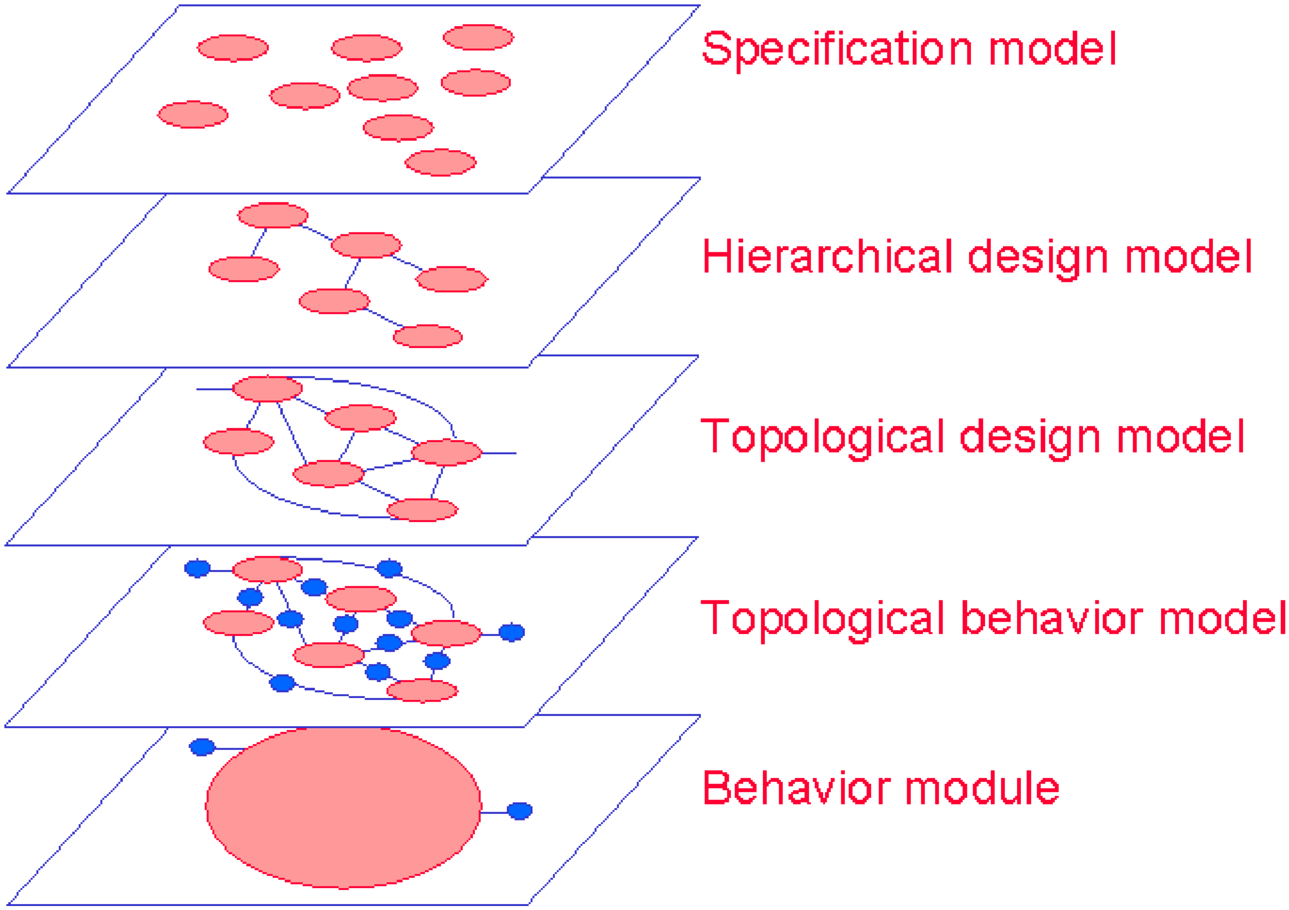
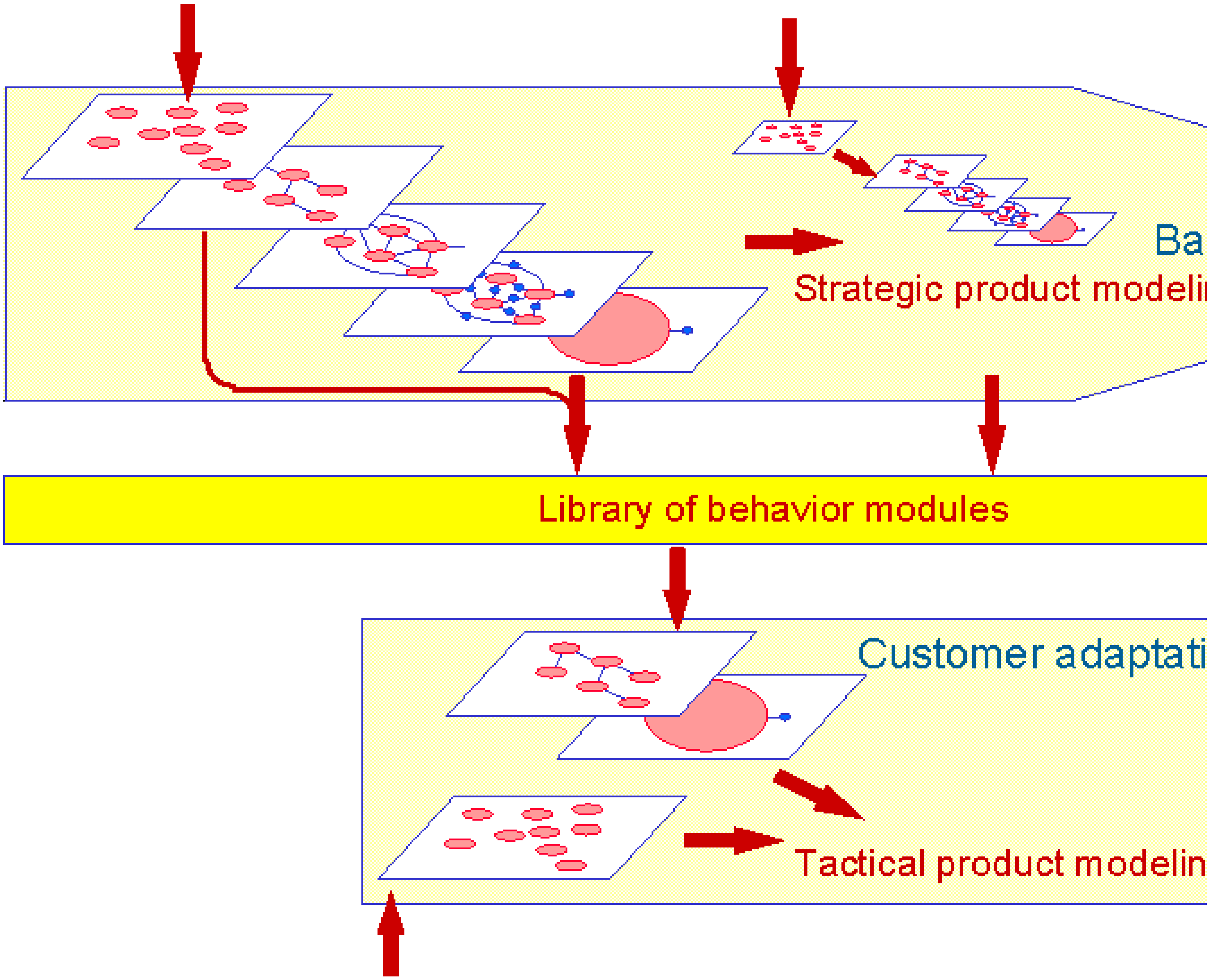


Figure 7. Constraints of the model



The results of FEM calculation

The critical parts of the beam are near the end where the design stress was 99% of the limit stress. To hold web of the beam in the beam direction was necessary. Any free deflection of the beam end led to the rapid decrease of the carrying capacity of the beam. The solved model had constrained ends at web and at flanges in the longitudinal direction (Y). The flanges were also supported in the vertical (Z) and side direction (X). The stability of the upper flange is ensured on the real structure by the one side diagonal angles. The lower and upper flange on one side of the beam was constrained. The model shown collapse at the compressed part of the undulated web and high stresses were found at the one-side welds and their vicinity. At all cases the collapse of the web occurred near the area of stresses 210 MPA. The values of obtained stresses were critical for the oneside welds, which have smaller carrying capacity then ordinary welding (Fig.8). The effect of the eccentricity of the load (position in one quarter of the flange width) was not significant for the critical part of the beam. The eccentricity led to the different distribution of stresses in the middle part of the undulated web (Fig.9). The behavior of the model was highly dependent on the supports of the studied beam. The other parts of the structure support the critical area of the web longitudinally. The model constraints were assumed to be rigid. The actual structure is more flexible; therefore the stresses at the real structure should be higher. The stresses closed to the presumed design yield point occurred also in the one-side weld area. The additional welding of the imperfect sections is therefore fully justified. To compare effect of one-side welding and the ordinary welding on the beam with the undulated web the other model was solved (Fig. 10). The more uniform distribution of stresses was obtained. No higher stresses occurred at the area of welds except at the critical part of the beam. In all these cases the accidental use of the steel with the higher design stress was substantial for the behavior of beams. Several stresses were compared. The stresses perpendicular to the direction of weld were most critical. The nonlinear deflections of the imperfect web at this region were bigger as the supporting one-side weld was softer then ordinary both side welding. The results of described FEM study of the beam with the undulated web caused by the improper welding corresponds to the theoretical study (Braham, M., Maquoi, R., Rangelov, N., Richard, C., 1995). The effect of the undulation of the web is in the range of several percents. However this imperfection can be dangerous for the structure under certain other conditions. The structure that is designed up to the design limit or very close to has critical areas that can collapse under the design load. The one-side welds are one of the critical regions. In the case of a weak design and production the technology of one-side welding should not be provided. The steel structure designed up to the upper limit of the design stress should be very carefully designed and fabricated. All kinds of imperfections are adding some additional stresses and one-side welds are sensitive to this. The nonlinear behavior of the imperfect beam is very important. Only the detail modeling can sometimes help to solve the situation.

Figure 8. The critical Z stresses at the web

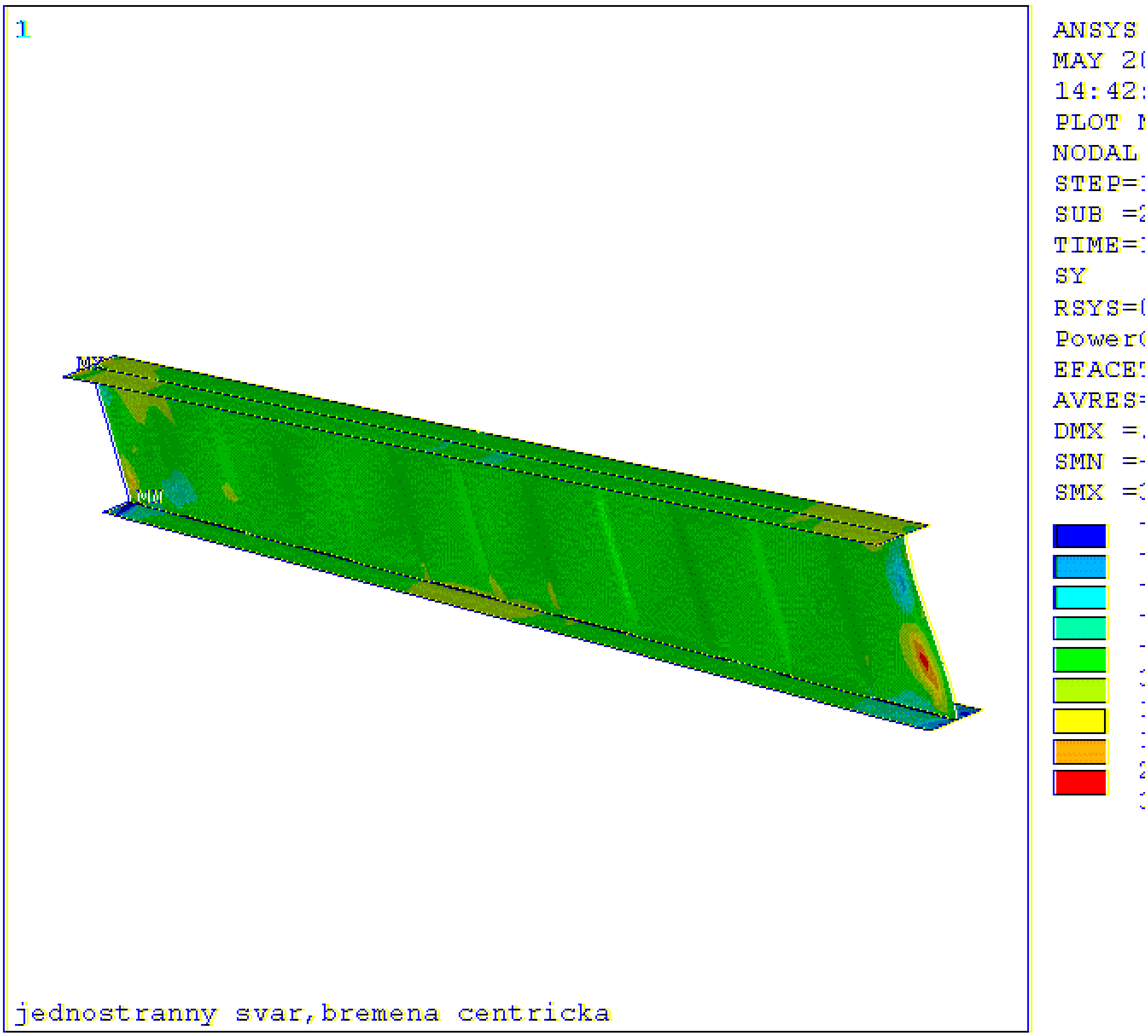


Figure 9. The beam eccentrically loaded-Z stresses

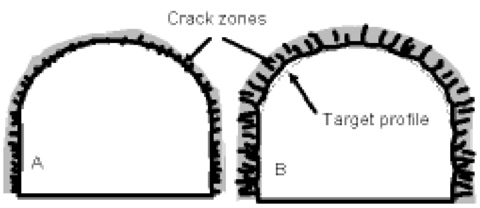


Figure 10. Both side welding - Y stresses

THE BEHAVIOR OF I WT GIRDERS WITH UNDULATED WEBS AND ONE-SIDE WELDS

The description of the system

One of modern structural type of beam members is a one-side welded beam with the web made from corrugated (undulated) metal sheet. The I beams are welded automatically on one side only. The web is relatively thin (2mm only) and flanges are 250x20 mm thick. The undulation of the web is 40mm. The height of tested and calculated beams was 625mm.The beams are light, stability of the web seems to be good. The dissertation thesis (Novak, R., 1999) studied the behavior of these beams. Experimental data and FEM calculation were compared and calibrated. The possibility to use this type of beam for the craneways was mentioned too. The problem in focus is the stability of the web and the carrying capacity of the beam. The research based on experimental testing and

FEM calculation was provided. The 18 samples were tested. The test specimens were 1000mm long beams simply supported and loaded in the middle by the vertical force. At the ends the beams the vertical stiffeners of the web were added. The beams were supported against lateral buckling in the upper third of the height at the end vertical supports. The influence of the load eccentricity was tested. The basic eccentricity was chosen as a half of the wave of the undulated web, that is 20mm. The option of large deflection was used, not the stress stiffening. The trilinear relation for the web material was used, to follow more accurately the nonlinear behavior of the slender web. The imperfections of the web, which are very important for their behavior is rather difficult to measure. For the model was at the dissertation introduced the shape with affinity to the shapes measured at the tests. Maximum imperfection was from 0.5 to 1.0 mm. Each tested sample was solved by FEM. Each model had about 1700 elements. The solution was done on the PC. The SHELL43 elements were used for the beam. For the crane rail the SOLID45 was used. The contact between the crane rail and the upper flange of the beam was modeled by CONTAC52 elements. Only the symmetrical half of the beam was modeled. For the steel at the flanges and end stiffeners and the bilinear curve for the steel was used.

Figure 11. The geometry and restraints of the WT beam model

Figure 12. The stresses at the beam

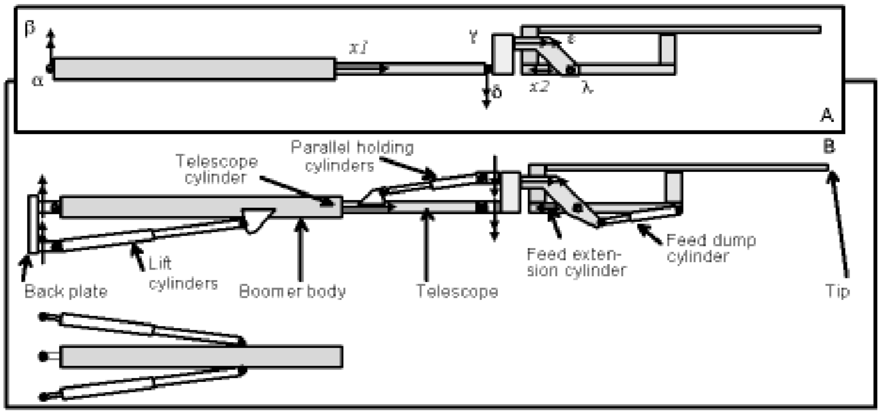
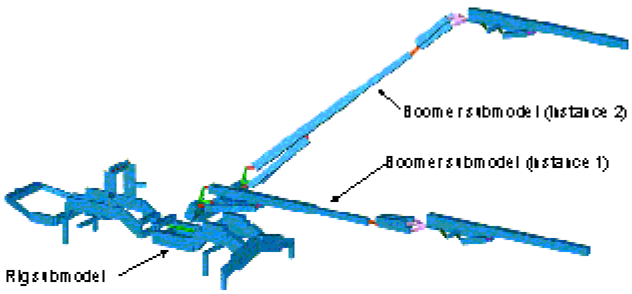


Figure 13. The deformed web



The FEM model of the beam and experiments

The results of FEM analysis and experiments. The results of experiments were carefully compared with the FEM results. The coincidence was good. The comparison of principal stresses at three location of the beam is at the table 4.1. The maximum calculated limit load varies from the experimentally received value within the range 2%. Only in one case was difference 85. The effect of the eccentricity of the load, which was very interesting, was not substantial. There was no difference observed on the carrying capacity of the beam. The one-side welding has also no effect on the behavior of the beam. Base on the statistic evaluation the final formula for the calculation of the maximum load for the web with respect to the crane rail used.

$$F_{Rk} = (78.9 * t_w + 3.2 * t_f - 14.7) * \sqrt[3]{\frac{I_f + I_R}{I_f + I_R^*}} \quad (4/1)$$

where

Ik is moment of inertia of the crane rail with respect to the self axis of inertia,

Ij is moment of inertia of the flange with respect to the self axis of inertia,

tw,, tf are thickness of the web and flange in mm.

The experience with the FEM solution.

The FEM solution of the structural system shown very closed agreement with experiments. The analytical solution of the system with geometric imperfections and with material nonlinearity is practically impossible The final result of the combined experimental and FEM solution is the formula that allows calculating carrying capacity of the undulated web of the beam immediately.

THE GEOMETRICALLY NONLINEAR BEHAVIOR OF BEAMS

The general equation for 3D beam for the small strain

The FEM analysis allows solving rather complicated relations. The nonlinear behavior of the 3D beam is usually using the well known formula with stress stiffening

S = (k E + kG) U (5.1)

Where the S is the load vector, U is the joint deflection vector and k E and kG are the elastic and geometric stiffness matrix. The derivation of the equation (5.1) is based on the following assumptions:

1. 3D members are straight without any imperfections
2. The local coordinate system of the member follows the right hand rule and is coincident with major principal axis of the member
3. Navier's hypothesis is valid for the cross section of the member.
4. Torsion is assumed to be Saint Venant type, i. g. warping is neglected.
5. The load step increment is finite and load is constant during the load step.

In these derivation are omitted higher order terms. If we use all the terms of the energy due to the axial deformation and don't assume the constant load within the increment, we are receiving more complex equation as follows for the 3D bar can be written as

$S_i = \mathbf{k_{Ei}}\ \mathbf{U} + \mathbf{(U^T\ h_i\ U)} + u_j\ \mathbf{(U^T\ q\ U)} + u_j\ \mathbf{(U^T\ g\ U)}$ (5.2)

and for the 3D beam elements can be written as

$S_i = \mathbf{k_{Ei}}\ \mathbf{U} + \mathbf{(U^T\ {}^7h_i\ U)} + \mathbf{(U^T\ {}^1h_i\ U)} +$

$u_i\ \mathbf{(U^T\ {}^ie1\ U)} + u_{i+6}\ \mathbf{(U^T\ {}^{i+6}\ e2\ U)} +$

$u_j\ \mathbf{(U^T\ {}^je3\ U)} + u_{j+6}\ (\mathbf{U^T}\ {}^{j+6}\ \mathbf{e4\ U})$ (5.3)

where both equation can be simplified to the form

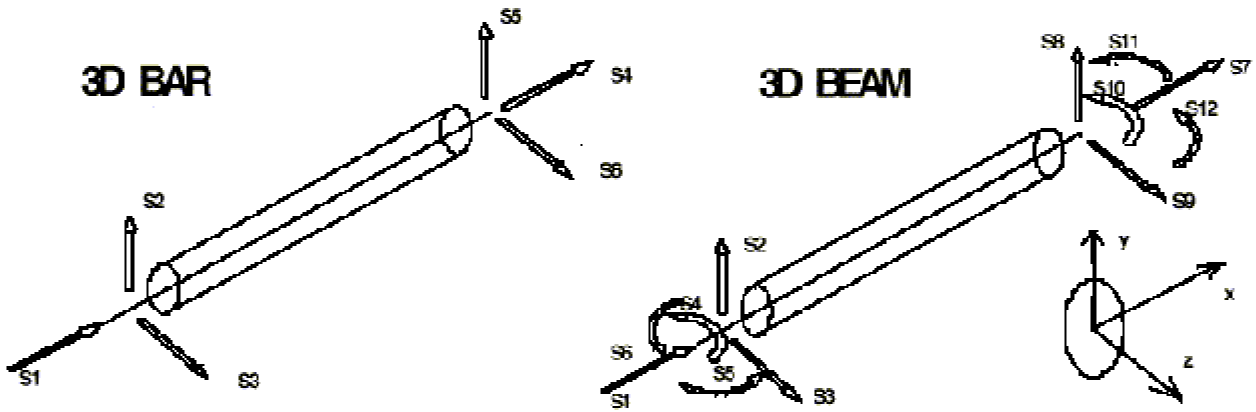
S = SE +SG + SQ (5.4)

The equation (5.4) separates the influences of different deflections during the nonlinear solution in very clear manner. Detail derivation and the description of these equations are given by Vašek a) (1998). The equations show that the nonlinear behavior of the beam is dependent on cross sectional properties that definition is given at following. The new cross sectional properties are derived for the nonlinear part in the form

$$K_{,} = \int_A z^4\,dA,\quad K_{,} = \int_A y^4\,dA,\quad K_{,} = \int_A z^2 y^2\,dA, \qquad (5.4)$$

The stiffness matrices $\mathbf{h}_i$, $\mathbf{q}_s$, $\mathbf{g}$, ${}^7\mathbf{h}_i$, ${}^1\mathbf{h}_i$, ${}^i\mathbf{e}_1$, ${}^{i+6}\mathbf{e}_2$, ${}^j\mathbf{e}_3$, ${}^{j+6}\mathbf{e}_4$ at the equation (5.3) and (5.4) are pre and post multiplied by the vectors of all joint deflections. This is the representation of the nonlinear effect. The deflections perpendicular to the bar or beam axis are premultiplying each nonlinear part as governing effect on the nonlinear deflection at the joint. These effects can be adopted for the stability and/or nonlinear solution of the space structural frameworks. The effects of semirigid joint can be also involved into that solution as a part of basic stiffness matrix of the system. The coefficients that express the level of rigidity of the particular joint can be calculated by FEM analysis and calibrated by an appropriate experiment.

Figure 14. The 3D beam element with 12 degree of the freedom -- small strains



CONCLUSION

The FEM studies described above brought very valuable results of the solution of some complex problems. All the problems studied were geometrically and materially nonlinear. We can distinguish two different groups of the civil engineering problems. At the first group is governing question of contact of any structural part and the distribution of plastic zones in the studied part. The material constants for contact elements are necessary to compare with the experiment. The results were dependent on the mesh density and value of the material constant. The final acceptable solution is usually limited by the admissible deflection of the part with stresses at the yield point. These limitations are nowhere given and should be defined. The good tool for this definition is the deformation energy spent over the deformed region of the structural part. The energy can be calculated by FEM program, e.g. ANSYS. The definition of admissible deflection is highly dependent on the geometry and topology and structural configuration of the system. In the case of space semirigid joint is the energy spent on the deformation composed from the axial and shear part. The axial part comes from the axial and bending and warping deformation, the spent shear energy comes from the shear and due to the torsion of the part of the joint. We should define the semirigidity of the joint with respect to all six or seven (the bimoment) general forces in space. The coincidence with experiments was good in the majority of cases especially at the elastic range of stresses, which is possible to check by strain gages. The solution of thinwalled structures is dealing mainly with imperfections and local or over all stability of the systems. The plastic regions are usually developed at buckled parts of structure. The values of imperfections is affecting the behavior of the systems and the proper model is important. Purely analytical solution of similar cases doesn't gives a good answer for the real problem. The FEM analysis is becoming more important for these complex problems with the real imperfect shape.

ACKNOWLEDGEMENTS

The FEM studies described above were enabled thanks to the partial support of the CVUT research actions Z3: Pocítacova simulace J04/98: 210000003 and Z1: Funkcní zpusobilost a optimalizace stavebních konstrukcí) J04/98: 210000001, which support financially the research in the field.

REFERENCES

Braham, M., Maquoi, R.,Rangelov, N., Richard, C.: „L'Influence des Defauts de Planeite deL'Ame des

Pro-files Reconstitues Soudes sur Leur Resistance en Flexion at Compression", Construction Metallique, No.1-1995

Novak, R.: „The Use of Girders with Undulating Webs for Crane Runways", Doctoral Thesis, CTU in Prague, 1999

Vasek, M.: „The Non-linear Behavior of Large Space Bar and Beam Structures ", G.A.R. Parke and .M.Howard, Space Structures 4 the Conference proceedings on Space Structures, 665-674, Thomas Telford Services Ltd. Lon-don, (1993)

Vasek, M.. Drdacky M., Hoblik, K.: „ Research Report of the Czech Grant Office no.103/93/2027, Space Roof Structural System", Pittsburgh, Prague, (1996)

Vasek, M.: „Non-linear small strain separate effects solution for 3D bar system ", proceedings of the 4th World Congress on Computer Mechanics. CD-ROM. Buenos Aires, IACM. (1998)

Vasek, M.: „Small-strain non-linear relations for 3D space beam system", in proceedings Long-span and high-rise structures. Congress IABSE KOBE (1998)

Vasek, M.: „The Influence of Imperfections on the Be-havior of One-Side Welded Beams", in Proceedings Eu-rosteel, Vol.1, pp. 211-214, CD, CTU Praha (1999)

Vasek, M.: "The Structural Analysis and FEM Pro-grams as an Educational Tool for the Civil Engineering Stu-dents", Proceedings of Civil Engineering Learning Tech-nology "CELTIC" Cardiff 1999" str.81-88, editor R. M.Lloyd, C.J.Moore, Thomas Telford Ltd., London (1999).