Journal of Machine Engineering, Vol. 8, No. 3, 2008

workpiece, fixture
quasistatic analysis

Miroslaw GROCHOWSKI[1]
Jerzy JEDRZEJEWSKI[1]

COMPARISON OF TWO FEA-BASED APPROACHES IN PREDICTION OF WORKPIECE-FIXTURE STATIC BEHAVIOUR

Simulating of workpiece and fixture behaviour is commonly done with the use of Finite Element Analyses. In the most, if not all, cases researchers usually use traditional implicit integration scheme FEM codes (e.g. ANSYS, ABAQUS/Standard, NASTRAN, FEAP). In this paper a different approach is proposed. FEM code of ABAQUS/Explicit, based on explicit integration of equations of the motion is used to predict workpiece behaviour during alignment and fixing process in quasistatic state. Comparison of results obtained using both implicit and explicit techniques is also presented, results and differences are discussed.

1. INTRODUCTION

Large part of the assembly design in the aerospace and automotive industry, is based on thin-walled elements – optimised for the minimal mass. This comes from the need of reducing the energy absorption of products and achieving the most beneficial rating between the net weight and effective work of a given machine assembly. The need of precise machining and assembling such thin-walled, flexible elements forces the use of well-designed fixing devices for machining and assembly processes. When fixture is designed for very expensive workpieces, or when the mass scale production process takes place, well designed and optimised fixture is crucial for the production process. Precise prediction of the workpiece behaviour in fixturing process should result in fixture optimisation which results in better manufacturing accuracy.

One of early approaches of fixture-workpiece modelling is the rigid body approach – analysing of fixtures using a rigid workpiece model. Its advantage is a relative simplicity of mathematic formulation of the problem, but it is useless when the deflection of a workpiece is one of the points of interest. Rigid body approach is useful for contact force prediction analysis [12], kinematic analysis [1], or fixture planning analysis [13]. The another approach is the application of the Finite Element Method to predict workpiece

[1] Wrocław University of Technology, Institute of Production Engineering and Automation, ul. Łukasiewicza 5, 50-371 Wrocław, Poland, e-mail: miroslaw.grochowski@pwr.wroc.pl, jerzy.jedrzejewski@pwr.wroc.pl

behaviour under the influence of cutting forces [8,6,7] or for predicting workpiece-fixture loads under clamping and machining. In all published papers different contact formulations are used, different clamping schemes, as well as different workpiece shapes, but all the papers show results obtained using the traditional implicit integration technique for static state analyses [11,12,2], dynamic implicit technique for dynamic analyses [4] or both static and dynamic implicit analyses [9]. Interesting approach, the use of a pair of FEM solver and dynamic analysis software DADS, is presented in paper [5].

Rather than another work, in this paper an explicit integration technique is used to predict workpiece behaviour in quasistatic state. The comparison between static implicit analyses and explicit analysis in quasistatic state is also investigated. The same load scheme, contact and finite element mesh definition is used in all analyses. Differences are discussed, causes of them are suggested and some of them are discussed in details and computationally verified. The Abaqus/Standard and Abaqus/Explicit version 6.7 FEM codes are used for all presented analyses.

2. SHORT BACKGROUND OF USED FINITE ELEMENT ANALYSES

There are two main approaches connected to solving problems with the behaviour of solid bodies (as well as fluids, gases, magnetism, etc.) with the use of the Finite Element Method. In hereby chapter, these methods will be briefly discussed on the example of solving solid body mechanics problems.

First, and the most commonly used approach is the traditional implicit technique of integrating the equations describing the analysed system. Such approach, in relation to the static problem presented in hereby work, is based on the solution of a matrix equation system [14]:

$$[K]\{u\} = \{R\} - \{F\}_p - \{F\}_{\varepsilon 0} \tag{1}$$

In equation (1), left side is the product of a system stiffness matrix $[K]$ and a vector of nodal displacements $\{u\}$, while right side of the balance equation consists of external forces acting on the system $\{R\}$, nodal forces $\{F\}_p$ and remaining forces, such as thermal loads $\{F\}_{a0}$. In case of non-linear problems, the above equation system is multiply solvable with the use of the incremental procedure (incremental increase of force); for every increment, a convergent solution is looked for by consecutive iterations, providing it was not reached in the first attempt. The advantage of such method is the possibility of reaching the solution after relatively low number of increments and iterations. Whereas the disadvantages are: the need of time-consuming solving of large system of equations and the possibility of not reaching any solution in case of the appearance of serious problems with the convergence.

Another approach to solving the problems possible to describe by means of FEM is explicit solving of dynamics equations in time. The main assumption for such method consists in the integration of motion equations according to the following equations [3]:

$$[\dot{u}]_{(i+\frac{1}{2})} = [\dot{u}]_{(i-\frac{1}{2})} + \frac{\Delta t_{(i+1)} + \Delta t_{(i)}}{2}[\ddot{u}]_{(i)}, \tag{2}$$

$$[u]_{(i+1)} = [\dot{u}]_{(i)} + \Delta t_{(i+1)}[\dot{u}]_{(i+\frac{1}{2})}, \tag{3}$$

where $[u]$ is the displacement, $[\dot{u}]$ is speed, $[\ddot{u}]$ nodal acceleration and i is the time increment number. The accelerations matrix $[\ddot{u}]$ is described by the following equation:

$$[\ddot{u}] = [M]^{-1} \cdot \left([F]_{(i)} - [I]_{(i)}\right), \tag{4}$$

where $[M]$ is a point mass matrix, $[M]$ is a vector of external forces and $[I]$ is a vector of internal forces. Because of the diagonal form of a matrix $[M]$, the solving process of the above equations is very effective. Moreover, the presented equations produce convergent results, thanks to which it is possible to avoid the application of the iterative procedures (with the exception of the problems, in which dampers appear [3]). The effect of applying the explicit integration of motion equations is calculating the propagation of a sound wave in the analysed object. This requires the application of a very small time step, selected in such a way, so during one increment a wave of sound travels between the nodes of a smallest finite element in the analysed object. In case of applying the described method for quasistatic analyses, its main advantages are: no problem with reaching the convergence, high effectiveness in analysing the phenomena of multi-body contact, linear dependency of computational costs from the problem size, computer memory savings. Disadvantages are the need of using special loading schemes, significant increase of computation time taking place in case of a local concentration of the finite element mesh, high demand for the computer's computational power.

Reassuming, the traditional method of implicit integration is based on the search of the equilibrium conditions by solving the system of equations with many variables, based on a symmetrical stiffness matrix [K]. Such method is sensitive to the appearance of problems with reaching the convergent solution. A method of integrating the motion equations is based on solving the equation system based on a diagonal mass matrix [M], which in effect comes down to solving the equation system containing one variable each. Such method is low-sensitive to the appearance of problem with reaching the convergent solution, which predisposes it to the modelling of a multi-body contact.

3. FIXTURE-WORKPIECE DEFINITION

Fixture-Workpiece system arrangement presented in this paper is defined as an analytically rigid fixture equipped with linearly movable analytically rigid locators and clamps. The clamps and the locators interact with a solid deformable workpiece.

3.1. WORKPIECE DEFINITION

Workpiece is defined as a deformable 3D solid object (Fig. 1a). The workpiece is also defined as a simplified exemplary object commonly used in the industrial application. Its level of simplicity is dictated by the compromise between the ease of modelling and interpreting of the results and similarity to actual parts used in real applications.

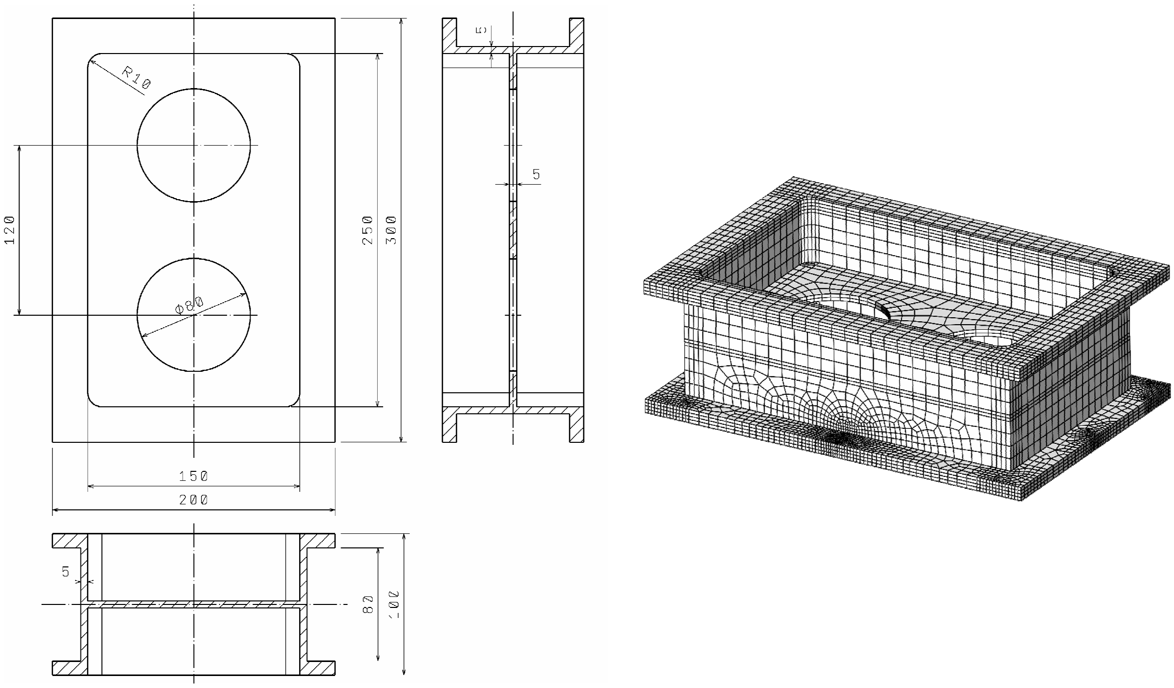

Fig. 1. Geometry (a) and Finite Element mesh (b) of the analysed workpiece

The analysed workpiece is defined using material properties similar to a typical aluminium alloy: *Young* modulus: E=6E10Pa, density ρ=2700 kg/m^3, *Poisson* ratio ν=0.3 and no plasticity. Mass of the part is approx 2.5kg. Finite Element mesh (Fig. 1b) is based on 17696 of nodes and 12510 of linear, reduced integration hexahedral elements with enhanced hourglass control (C3D8R). The reason for using first order elements is that Abaqus/Explicit uses only first order elements.

3.2. FIXTURE DEFINITION

Fixture (Fig. 2) is based on 12 analytical rigid surfaces which represent locators (1-12) placed in space by fixing selected degrees of freedom. Number of degrees of freedom of fixed locators (1-6) is 0. The axial degrees of freedom of the clamps and movable locators are free.

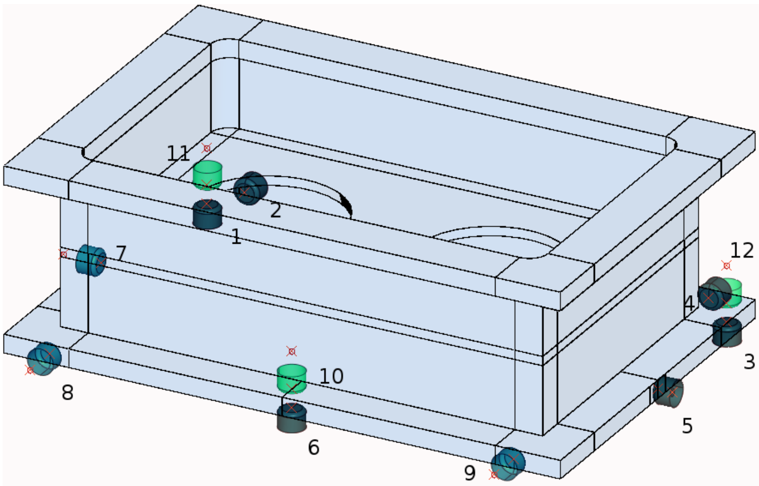

Fig. 2. Fixture definition. 1-6 – fixed locators, 7-9 – movable locators, 10-12 – clamps

In the movable locators (7-9) and the clamps (10-12), springs are defined at the active directions (Fig. 2). Stiffness values of the springs are K_1=2kN/m for the movable locators and K_2=200kN/m for the clamps.

3.3. CONTACT DEFINITION

The Abaqus/Standard and Abaqus/Explicit FEM codes use completely different contact algorithms (Section 0), but some of the contact properties could be defined by the user in the same way. In this subsection, contact definitions of all analysed models is presented in short.

Surface to surface discretization method and finite sliding tracking approach were used in all analyses using both static and quasistatic analyses. Penalty friction model with the Coulomb friction coefficient μ=0.1 is defined between all contacting surfaces in all analyses. Abaqus default "hard" contact behaviour is used at the normal direction, no initial clearance between contacting surfaces is used. Parameters which differ between analyses are shown in Table 1.

Table 1. Contact properties which differs between all analyses

Analysis Nr.	FEM solver	Mechanical Constraint Formulation	Sliding tolerance
S1	Abaqus/Standard	Penalty	0.005
S2	Abaqus/Standard	Penalty	0.0005
E1	Abaqus/Explicit	Penalty	NA (infinite stiffness)
E2	Abaqus/Explicit	Kinematic	NA (infinite stiffness)

3.4. ANALYSIS STEPS AND LOADS DEFFINITION

The analyses are divided into 6 load steps:

1. Applying gravity load (g=9.81 m/s^2); step time t=6µs.
2. Compressing spring of the movable locator (7) by 1mm which acts as the load of 2N; t=3µs
3. Compressing springs of the movable locators (8,9) by 1mm which acts as the load of 2N per locator; t=3µs.
4. Release all 3 springs, apply force F_1=50N per locator on locators 7,8,9; t=6µs.
5. Compress springs of 3 clamps (10-12) by 1mm which acts as the load of 200N per clamp; t=6µs.
6. Release springs of clamps and apply the clamping force F_2=1kN per clamp; t=6µs.

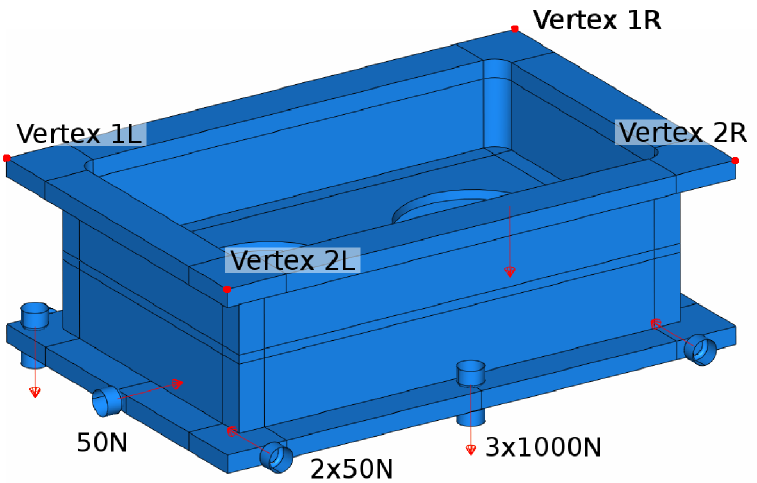

Fig. 3. Workpiece's load definition. a) static and quasistatic analyses, b) buckle analysis

In explicit, quasistatic procedure, the time period of each load step should be comparable with the cycle time of a first natural frequency of the analysed model. The first natural frequency of the analysed workpiece obtained using the Linear Perturbation procedure is about 425Hz so the largest cycle time of a free workpiece is about 0.002s. Conservative periods of each step were chosen: steps 1,4-6 take 0.006s per step, steps 2 and 3 take 0.003s per step. A special smooth load rate is used, for reducing the dynamic phenomena influence at the quasistatic state. In the smooth load rate first and second derivatives of the loads equal zero at the beginning and at the end of each step.

As no time dependent variables are defined in the analysed model (e.g. time-domain material properties) the step time has no influence on results obtained using Abaqus/Standard FEM code, but for better visualisation the same time period and the same load rate was chosen as in all analyses.

4. RESULTS OF ANALYSES

The displacement of top 4 vertices of the part (Fig. 3a) was analysed and presented below (Fig. 4 and 5). In all cases the most significant deformation is observed in the Z direction. The Z alignment load is relatively small (*F*=50N), but it causes significant deformation because it acts in direction of the largest flexibility of the workpiece-fixture set. Other significant displacement changes are observed in the last step of all analyses when high forces of 1000N act on 3 clamps. Two most interesting phenomena observed in the analyses are discussed in the next 2 paragraphs. There are two, more interesting phenomena which are discussed in next 2 paragraphs.

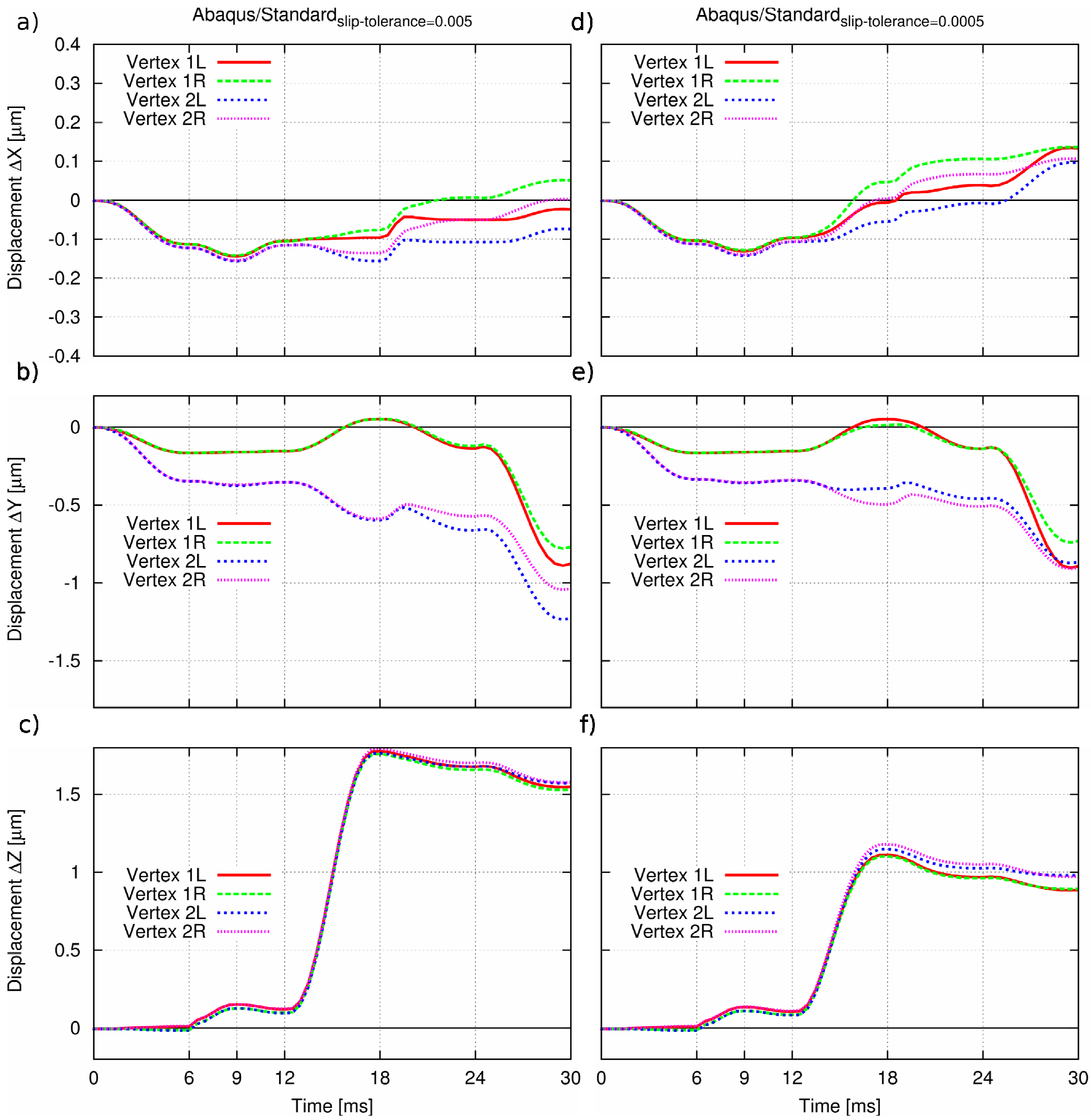

Fig. 4. Displacement of 4 top vertices obtained using Abaqus/Standard FEM code and different tangential slip rate tolerance (ticks at time axis of all presented plots represents steps described in section 3,4)

Displacements in the Z direction obtained in the S1, E1 and E2 (Table 1) analyses are similar, but displacements obtained in the S1 analysis are about 50% larger. Such difference is caused by wrong tangential behaviour of the contact model caused by too large value of the default slip rate tolerance of the characteristic surface dimension. The same value of deformation is observed in frictionless analysis. Reducing the slip rate by 10 times, from default 0.005 to 0.0005 of the characteristic surface dimension, reduces incorrect behaviour of the friction model (Fig. 4).

Second and the more interesting phenomenon is the untypical behaviour of vertices observed in the X and Y directions of the E2 analysis (Fig. 5).

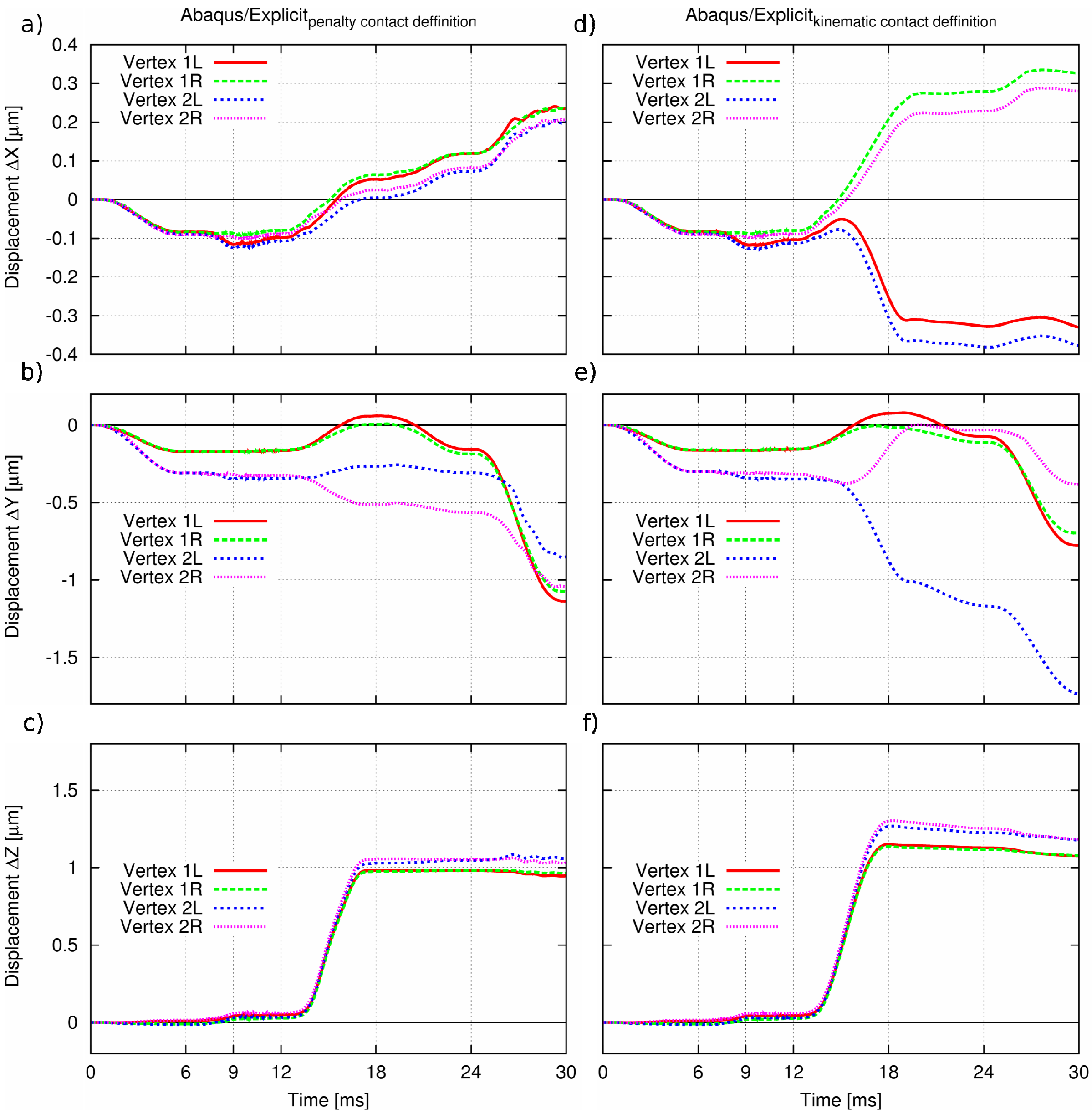

Fig. 5 Displacement of 4 top vertices obtained using Abaqus/Explicit FEM code and different contact definition (ticks at time axis of all presented plots represents steps described in section 3.4)

In the analyses S1, S2 and E1, magnitudes and forms of vertex displacements in the X and Y directions are comparable, but the results obtained in the E2 analysis are distinctly different. The application of the explicit integration method uncovers the possibility of twisting of the machined workpiece under the influence of acting forces. This is the behaviour of a model, which is not observed in the remaining cases. However, such phenomenon is exclusively observed when the kinematic contact definition is applied. The application of a penalty contact definition in case of quasistatic calculations with the use of the explicit method for integrating the motion equations also does not cause the appearance of the above mentioned phenomenon.

Fig. 6 presents a deformed form of the analysed machined workpiece in the X direction, obtained in the most characteristic simulation moment, after step no. 4 – applying the force of locators. In this group of graphs the results of S1 and E1 analyses were not shown, because the deformed form of the workpiece is in these cases almost identical to the form obtained in the S2 analysis, shown on Fig. 6a.

Fig. 6b and c present the deformed form of a workpiece achieved in the explicit quasistatic simulation and in additionally conducted static simulation, assuming that in the case c the friction coefficient between locators 7,8 and 9 and the machined workpiece amounts 0, while keeping the value of the friction coefficient for the remaining pairs of surfaces, equal to 0.1. The obtained deformation values are not identical, but the analogy between the results of both simulations is clearly visible. The difference of a friction coefficient is the obvious cause of the appearance of different deformation values, but it can be assumed that the following factors can also be responsible for the arisen difference:

1. Radically different mathematical description of the problem.
2. Numerical errors in the calculations.
3. Insufficient elimination of dynamic interactions taking place in the quasistatic analysis.

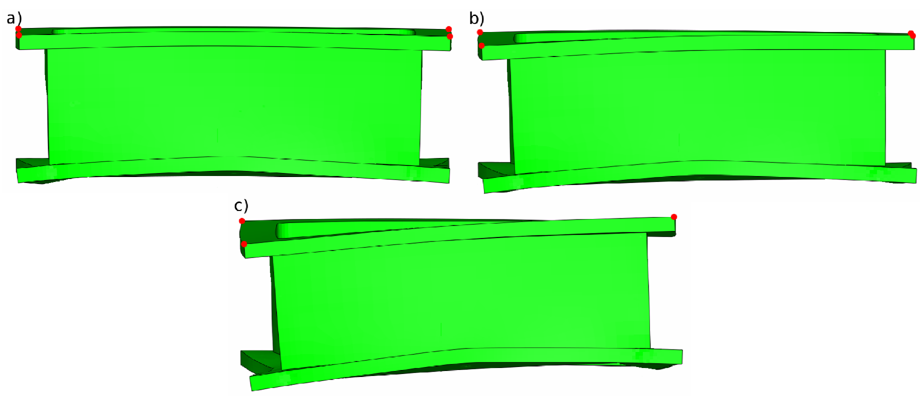

Fig. 6. Deformed shape of the analysed workpiece after applying the load from locators step (3rd) obtained in: a) static analysis, friction coefficient of 0.1 (Abaqus/Standard); b) quasistatic analysis (Abaqus/Explicit); c) static analysis, friction coefficient: 0.1, locators 7,8 and 9 - frictionless (Abaqus/Standard)

5. CONVERGENCE PROBLEMS

One of the main problems faced while solving problems with the use of FEM is the problem of the convergence of solution. Problems with achieving the convergence are especially strong when dealing with the contact problems. In order to make it possible to find the solution convergent to the correct solution, the FEM solver can be equipped with the functions facilitating the search for a convergent solution. In case of the software used by the authors, Abaqus/Standard, such functions are mainly automatic reduction of a time step realised after the detection of problems with convergence and an independent mechanism of searching for contact nodes. Unfortunately in case of detecting problems with convergence, the action of such mechanism leads to the significant decrease of time step, which can lead to the elongation of the computing time, even by few orders in relation to a convergent problem of the same size, and in some cases the achievement of a convergent result is simply impossible.

As an example, the authors conducted a trial of 11 calculations of a problem defined in the chapter 0, assuming varying friction coefficients and two values of the slipping tolerance (Table 2).

Table 2. Convergence of solutions for different contact properties.

Analysis No.	Friction coefficient	Sliding tolerance	Converged	Reason
0	0	NA	No (Step 4)	Too slow convergence rate
1	0.1	0.005	Yes	
2	0.1	0.0005	Yes	
3	0.2	0.005	No (Step 3)	Too many attempts
4	0.2	0.0005	No (Step 3)	Too many attempts; Too many SDI
5	0.3	0.005	No (Step 4)	Too slow convergence rate
6	0.3	0.0005	No (Step 3)	Too many attempts
7	0.4	0.005	No (Step 4)	Too slow convergence rate
8	0.4	0.0005	Yes	
9	0.5	0.005	Yes	
10	0.5	0.0005	Yes	

The above example shows that the achievement of a solution in case of multi-body contact with the use of a calculation technique based on solving equilibrium systems of equations can meet with so severe problems with reaching convergence that the solution might not be obtained in a reasonable time. In this moment it is worth noting that the application of parallel calculation techniques, in which the model is divided to domains solved by individual processors (MPI) has an additional negative influence on the achievement of convergence. Therefore the authors, in order to solve particular problems, used only one processor per task.

In case of solving the problem defined in section 0 with the use of explicit integration of dynamics equations offered by Abaqus/Explicit, the problem with reaching convergence does not exist. Independently from the size of the introduced friction coefficient the solution is always achievable, and the calculation time by definition is always approximately the same. Additionally, the parallelisation of a task to many processors does not cause significant differences of obtained results. It must be however noted that it is necessary to minimise the influence of inertial forces on the behaviour of the model.

6. CONCLUSIONS

On account of problems with achieving the convergence of calculations in case of the application of the inexplicit system integration method, obtaining results might be very difficult or simply impossible. Admittedly there are numerous methods, which can be used for better model tuning with the capabilities of a program, but such process is laborious, requires very individual approach to every problem, often causing multiple increase of calculation time and still does not guarantee the achievement of intended results. These methods were not described in hereby work.

A competitive approach, which utilises the method of explicit integrating the motion equations of the analysed model with the use of a quasistatic load model, is almost free from convergence problems. Such approach only in extreme cases does not lead to a convergent solution. At the same time the time required to achieve the final solution in a small degree depends on the nonlinearity level of the analysed problem. While in case of traditional static methods a small model change, e.g. change of the friction coefficient, can considerably change the calculation time or even make their finishing impossible, in case of the explicit integrating of the motion equation method the influence on the computation time is negligible.

Based on the workpiece-fixture system it was demonstrated that the explicit motion equation integrating method in the application to solving the quasistatic systems can be a very attractive tool for analysing of systems, in which the multi-body contact takes place.

ACKNOWLEDGEMENT

Hereby work was realised as a part of the AFFIX project (http://www.affix-ip.eu/), which is a part of the EU 6th.framework program. The work was supported by Polish Ministry of Science and Higher Education. Calculations have been carried out in Wroclaw Centre for Networking and Supercomputing (http://www.wcss.wroc.pl), grant No. 109.

REFERENCES

[1] ASADA H., By A.B., *Kinematics analysis of workpart fixturing for flexible assembly with automatically reconfigurable fixtures.*, IEEE Trans Rob Automat RA-1(2), 1985, 86-93.

[2] ASANTE J.N., *A combined contact elasticity and finite element-based model for contact load and pressure distribution calculation in a frictional workpiece-fixture system,* Int J Adv Manuf Technol 39, 2008, 578-588.

[3] Dassault Systmes, *Abaqus Theory Manual Version 6.7.* 2007.

[4] LI B., MELKOTE N. S., *Improved workpiece location accuracy through fixture layout optimization*, International Journal of Machine Tools & Manufacture 39, 1999, 871-883.

[5] LIAO Y.J.G., HU S.J., *Flexible multibody dynamics based fixture-workpiece analysis model for fixturing stability,* International Journal of Machine Tools & Manufacture 40, 2000, 343-362.

[6] RATCHEV S. , GOVENDER E., NIKOV S., PHUAH K., TSIKLOS G., *Force and deflection modelling in milling of low-rigidity complex parts*, Journal of Materials Processing Technology 143-144, 2003, 796-801.

[7] RATCHEV S., HUANG W., LIU S., BECKER A.A., *Modelling and simulation environment for machining of low-rigidity components*, Journal of Materials Processing Technology 153-154, 2004, 67-73.

[8] RATCHEV S., LIU S., HUANG W., BECKER A.A., *A flexible force model for end milling of low-rigidity parts,* Journal of Materials Processing Technology 153-154, 2004, 134-138.

[9] RATCHEV S., PHUAH K. , LÄMMEL G., HUANG G., *An experimental investigation of fixture-workpiece contact behaviour for the dynamic simulation of complex fixture-workpiece systems*, Journal of Materials Processing Technology 164-165, 2005, 1597-1606.

[10] RATCHEV S., PHUAH K., LIU S., *FEA-based methodology for the prediction of part-fixture behaviour and its applications*, Journal of Materials Processing Technology 191, 2007, 260-264.

[11] SIEBENALER S.P., MELKOTE S.N., *Prediction of workpiece deformation in a fixture system using the finite element method,* International Journal of Machine Tools & Manufacture 46, 2006, 51-58.

[12] WANG M.Y., PELINESCU D.M., *Contact force prediction and force closure analysis of fixtured workpiece with friction,* ASME Journal of Manufacturing Science and Engineering 125 (2), 2003, 325-332

[13] WU Y., GAO S., CHEN Z., *Automated modular fixture planning based on linkage mechanism theory,* Robotics and Computer-Integrated Manufacturing 24, 2008, 38-49

[14] ZIENKIEWICZ, O.C., *Metoda Elementów Skoczonych* (in polish, translated from: *Finite element method in engineering science),* Warszawa: Arkady, 1972.