Transactions of the VŠB – Technical University of Ostrava, Mechanical Series
No. 1, 2009, vol. LV
article No. 1644

Radek ČADA*, Barbora FRODLOVÁ**

ANALYSIS OF ELEMENTS NETWORK INFLUENCE UPON SIMULATION RESULTS IN THE DYNAFORM 5.2 SOFTWARE

ANALÝZA VLIVU SÍTĚ ELEMENTŮ NA VÝSLEDKY SIMULACE V PROGRAMU DYNAFORM 5.2

Abstract

The contribution concerns creation of elements network which is necessary at flat forming simulation and the analysis of meshing infuence upon simulation results. Meshing is an important process carried out before simulation, which affects final result. This process is analysed in the contribution, individual steps leading to aim are described. The possibilities of meshing in Dynaform 5.2 software evolved by American company Engineering Technology Associates, Inc. (ETA) are described. In contribution the meshing analyses are carried out on blank model determined by maximum shear stress trajectories method which is optimal for drawing of one of the representatives of intricate shape stampings from thin deep-drawing sheet-metal DC04 – left cover of ventilator for truck Tatra 815.

Abstrakt

Článek se týká vytváření sítě elementů potřebné při simulaci plošného tváření a analýzou jejího vlivu na výsledky simulace. Síťování je důležitý proces prováděný před započetím samotné simulace, který ovlivňuje její konečný výsledek. V článku je tento proces rozebrán, jsou popsány jednotlivé kroky vedoucí k cíli. Jsou popsány možnosti síťování v programu Dynaform 5.2, vyvinutém v americkou společností Engineering Technology Associates, Inc. (ETA). Analýzy síťování jsou v článku provedeny na modelu přístřihu stanoveného metodou využívající trajektorií maximálních smykových napětí, který je optimální pro tažení jednoho z představitelů výtažků nepravidelného tvaru z tenkého hlubokotažného plechu DC04 – levého krytu ventilátoru pro nákladní automobil Tatra 815.

1 INTRODUCTION

Computers and simulation software are more and more used in engineering production. Many programmes exist on base of finite elements method. For flat forming of sheet-metal the software Dynaform 5.2 is it for example. Absolute advantage of simulation is the fact that various parameters of tools for various types of products can be set and test without requirement of making expensive prototype, which is financially intensive and time-consuming for company.

In order the simulation results mostly draw near to real object, the meshing must be optimal for given stamping and given drawing process. All edge conditions must be dated up to software, like shape and size of the blank, motion and travel of the tools, force of the blankholder, velocity of the punch, material properties of sheet-metal and friction coefficient [2, 3, 4, 5].

* prof. Ing. CSc., VŠB-Technical University of Ostrava, Faculty of Mechanical Engineering, Department of Mechanical Technology, 17. listopadu 15, 708 33 Ostrava-Poruba, Czech Republic, tel.: +420 59 7323289, fax: +420 59 6916490, e-mail: radek.cada@vsb.cz

** Ing., VŠB-Technical University of Ostrava, Faculty of Mechanical Engineering, Department of Mechanical Technology, 17. listopadu 15, 708 33 Ostrava-Poruba, Czech Republic, tel.: +420 59 7323289, fax: +420 59 6916490, e-mail: barbora.frodlova.st@seznam.cz

The method of making mesh is different when meshing tool parts models then when meshing blank model. Different shape of elements and different size of elements are chosen which will be described in following chapters. Making model of stamping in CAD computer programme CATIA (this model will be used for making of tool parts models) is the first step. It must be saved like iges file and consequently it may be imported to Dynaform software. Model contains only lines and surfaces, the elements network must be created in Dynaform software.

2 MESH OF ELEMENTS ON MODEL

The mesh presents system of partition of area (tools, blank) on partial each other connected 2D cells (elements) in two-dimensional space. This network is a base of computation. There are additional rules:

- the analysis is more exacting the more elements are on model,
- the analysis is more exacting, the more smaller are these elements,
- the size of elements of tools effects the final shape of stamping, but accuracy of analysis the size doesn´t effect,
- the size of elements of the blank effects the accuracy and quality of analysis,
- the more exacting is the analysis, the more elongating is the time of analysis.

The way to making meshing is in module „Preprocess" selection „Element" → „Surface Mesh" in Dynaform software. The offer of three different forms of meshing and shape of elements will appear. For choice are „Tool Mesh" (suitable for tools), „Part Mesh" and „Triangle Mesh".

One of the main limiting factors is number of elements, which may affect quality of analysis and analysis time. The number of elements depends on size of elements, it must not be too small but neither too large. The user will enter the desired size of elements („Size"). Because the tools and also the blank are shape surfaces, the elements on this complex shape of surface not have to be the same. The user can this problem control in parameters of meshing („Mesh Quality") in software Dynaform, where the user can choose the maximum element size and the minimum element size („Max. Element Size", „Min. Element Size").

Quality mesh consists of following, geometrically regular, approximately the same and regularly distributed elements on whole surface. The shapes of elements will be combined while meshing the irregular surfaces. The software wants offer the best possible mesh for analysis to user. The situation maybe come, when optimum quad shape of elements („Quad") and less positive triangle shape of elements („Triangle") will appear on one meshing surface.

3 INTRICATE SHAPE STAMPING CHOSEN FOR ANALYSES

For evaluation of elements network influence upon simulation results in Dynaform 5.2 software the stamping of ventilator left cover (Fig. 1 and 2) which is a part of air line system of truck Tatra 815 was chosen by the authors like example of intricate shape stampings. This stamping is produced in firm Tawesco, Ltd., which is a daughter company of joint-stock company TATRA, Kopřivnice.

Fig. 1 Stampings of right (up) and left (down) cover of ventilator

Fig. 2 Ventilator left cover

For production of ventilator left cover the steel DC04 (11 305.21) is used which is mostly taken from VSŽ Oceľ, Ltd., Košice. The material is killed, non ageing and with very good properties for deep drawing. During its working the anisotropy of mechanical properties must be taken into account. Sheet-metal from steel DC04 is usually delivered recrystallizationaly annealed and additionally light cold re-rolled (marking .21 after numerical steel symbol).

Like initial blank the thin sheet-metal in plates with dimensions (0,9 × 1000 – 2000) mm ČSN 42 6312.32 from steel DC04 is used. Drawing of this stamping is carried out at crank presses PKZZ I 315 of German firm ERFURT, or at drawing crank presses LKT 250-A (firm Šmeral, Czech Republic). Anually about 2500 pieces is produced, correct number depends at number of ordered automobiles. During shape drawing operation at stampings often defects arises – secondary wrinkling and cracks.

4 MESHING OF DRAWING TOOL PARTS MODELS

For meshing of drawing tool parts models (punch, die, blankholder) the meshing form „Tool Mesh, Connected“ (it will connect the adjacent surfaces if the gaps are within the gap tolerance) was used.

In offer of parameters incoming mesh „Parameters“ (see Fig. 3) are chosen:

- Max. Size, Min. Size (maximum and minimum size of elements),
- Chordal Deviation (controls the number of elements at the radius),
- Angle (controls the inclination of the adjacent elements),
- Gap Tolerance (controls contact between two adjacent elements).

The maximum size of elements for tools was chosen 10. Others parameters were chosen by Dynaform software and because they were suitable, they were not changed. Then the part (tool like punch, die and blankholder) was selected („Select Surface“) and this selection was confirmed („Apply“). If the user is content with the meshing, he answers „Yes“ for the question „Accept Mesh?“. The elements network of punch model is on Fig. 4, the elements network of die model is on Fig. 5 and the elements network of blankholder model is on Fig. 6.

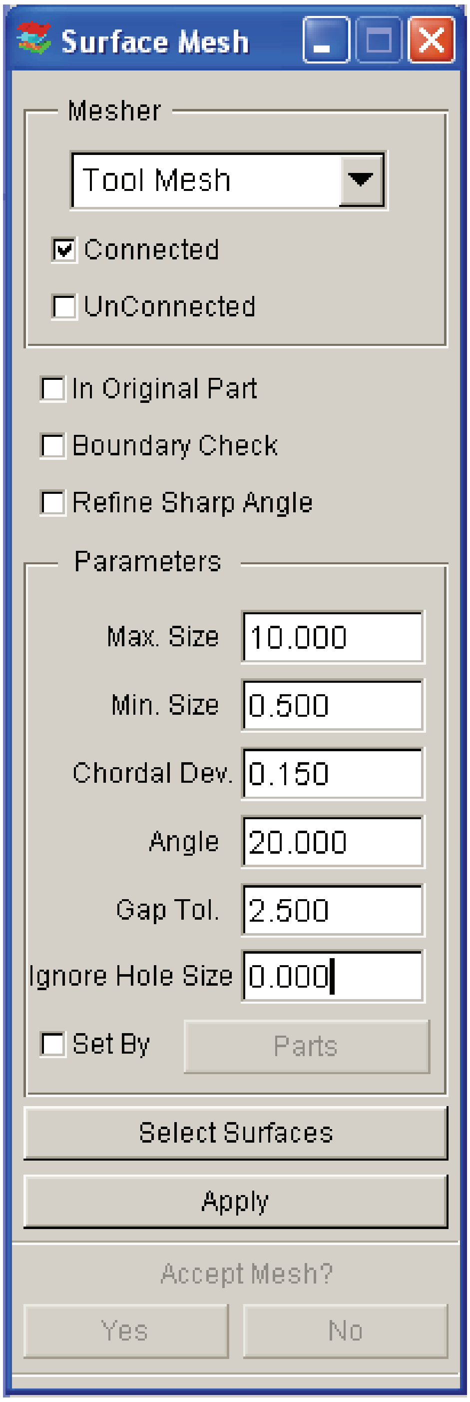


Fig. 3 Setting of surface mesh parameters in Dynaform 5.2 software

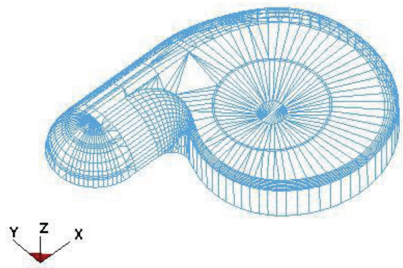


Fig. 4 Elements network of punch model

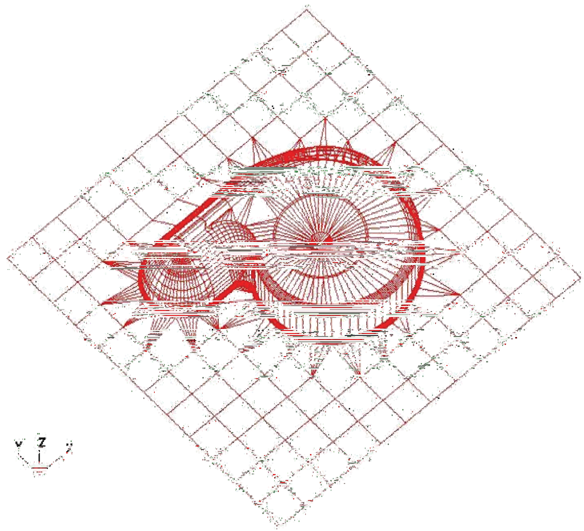


Fig. 5 Elements network of die model

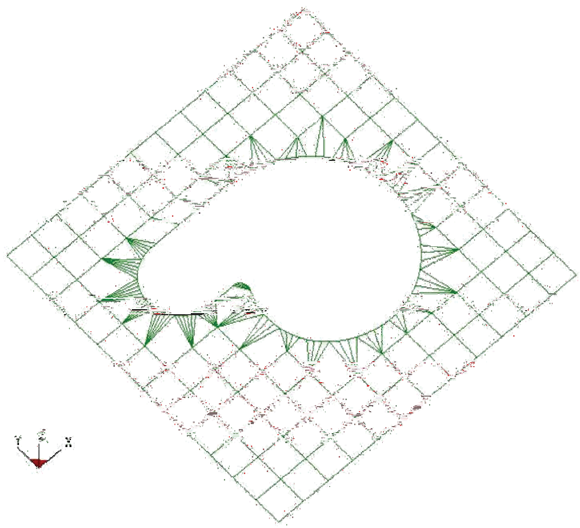


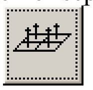
Fig. 6 Elements network of blankholder model

5 CHECKING OF DESIGNED MESH

Software always has an endeavour to making the best possible mesh, but mistakes can appear in mesh. The user must these mistakes find out and remove before the final simulation. Defects of mesh can make some problems and wrong result of simulation in final analysis.

The necessary steps for checking the mesh are on the list in module „Preprocess“ → „Model Check“. It is necessary to check the normals direction of separate model elements („Auto Plate Normal“, „Plate Normal“), which must have the same direction on all elements, check of continuity of model boundary line („Boundary Display“) and check of double and overlap model elements („Overlap Element“) first of all. These functions will be demonstrated on punch model (see Fig. 4).

5.1 Checking of the normals direction of separate model elements

Checking of the normals direction of separate model elements in Dynaform software is carried out by function „Auto Plate Normal“  . If the normal directions are different, they must be changed to the consistent (Fig. 7). If the user agrees with the direction, he says „Yes“ for the Dynaform´s question about acceptable orientation.

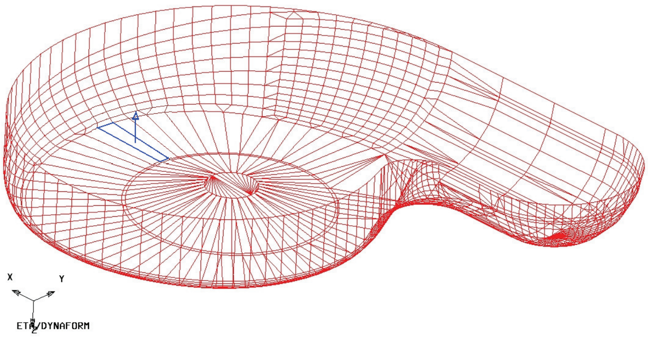
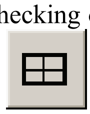


Fig. 7 Checking of the normals direction of separate model elements

Function „Plate normal“ does not show direction of normals on a model, but it draws boundary line between elements with reverse orientation. If the reverse directions of normals aren´t there, software promptly reacts in window „Prompt Area“ and it says: “Normal check completed. Normal is consistent“.

5.2 Checking of model boundary line

Checking of model boundary line in Dynaform software is carried out by function „Boundary Display“  . This function is useful for controlling the right contact of elements, because the mesh on the model must not contain a gaps and a mistakes. Fig. 8 shows superior punch model boundary line.

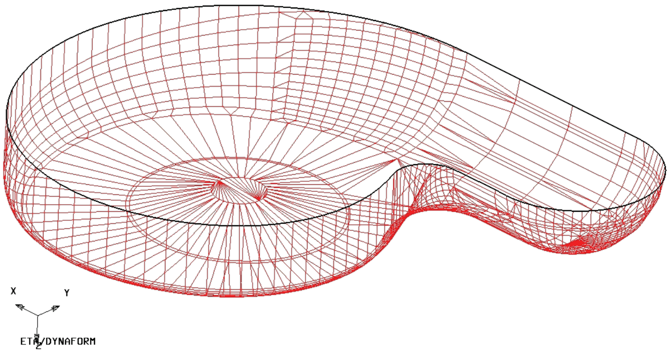
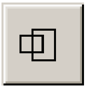


Fig. 8 Checking of model boundary line

5.3 Checking of double and overlap model elements

Checking and find of double and overlap model elements in Dynaform software is carried out by function „Overlap Element“ . If such elements are in model, it is possible to erase them. If the software does not find such elements, it promptly reacts by sentence: „No overlap element found“.

6 MESHING OF BLANK MODEL

For production of the left cover of the ventilator for truck TATRA 815 these blanks can be used: current blank, blank determined by method using the Czech State Standards, blank determined by method of sections, blank determined by method using maximum shear stress trajectories and blank determined by method using BSE module in Dynaform 5.2 software. Various meshing of blank determined by method using maximum shear stress trajectories (optimal blank for the left cover of the ventilator) are compared in next chapters with the effects of different meshing upon simulation result.

These forms of meshing in Dynaform were used for meshing the blank: „Tool Mesh“ and „Part Mesh“. The aim was discovery, which form of meshing is optimal for blank in term of accuracy final results.

6.1 Meshing of blank model by function „Tool Mesh“

The form of meshing „Tool Mesh, Connected“ was used (it will connect the adjacent surfaces in the gaps are within the gap tolerance).

In offer of parameters incoming mesh („Parameters“) are chosen: „Max. Size, Min. Size“ (controls maximum and minimum size of elements), „Chordal Deviation“ (controls the number of elements at the radius), „Angle“ (controls the inclination of the adjacent elements), „Gap Tol. “ (controls contact between two adjacent elements).

The maximum size of elements for tools was chosen 10 and 5. Others parameters were chosen by Dynaform software and because they were suitable, they were not changed. Then the part (blank) was selected („Select Surface“) and this selection was confirmed („Apply“). If the user is content with the meshing, he answers „Yes“ for the question „Accept Mesh?“. The mesh of elements for blank with element size 10 is on the Fig. 9, the mesh of elements for blank with element size 5 is on Fig. 10.

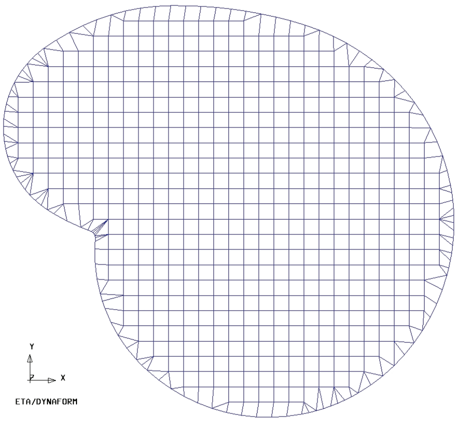


Fig. 9 Elements network of blank model created by usement of function “Tool Mesh” and setting of element size 10

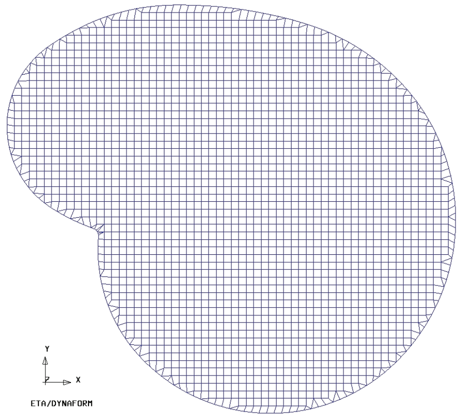


Fig. 10 Elements network of blank model created by usement of function “Tool Mesh” and setting of element size 5

6.2 Meshing of blank model by function „Part Mesh“

In offer of parameters incoming mesh („Parameters“) is chosen required size of elements („Size“), which was chosen 10 and 5. Others requirements for mesh are chosen in offer „Mesh Quality“, for example maximum and minimum acceptable size of elements (it was chosen: max. size 15, min. size 2). Others parameters were automatically chosen by Dynaform 5.2 software and because they were suitable, they were not changed. Following parameters were chosen too:

- „Boundary Check“ for control continuity boundary line of blank,
- „Check Surface“ for control surfaces,
- „Auto Repair“ for automatic repair of mesh.

Then the part (blank) was selected („Select Surface“) and this selection was confirmed („Apply“). If the user is content with the meshing, he answers „Yes“ to the question „Accept Mesh?“. The elements network of blank model with element size 10 is on the Fig. 11, the elements network for blank model with element size 5 is on the Fig. 12.

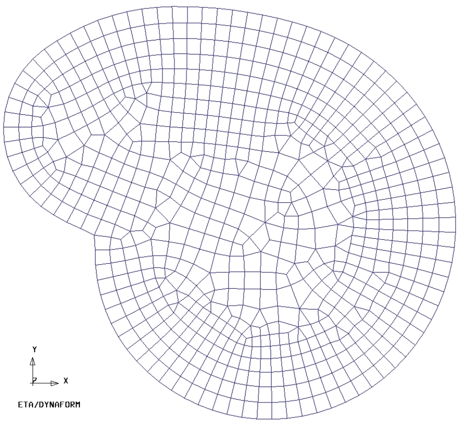


Fig. 11 Elements network of blank model created by usement of function “Part Mesh” and setting of element size 10

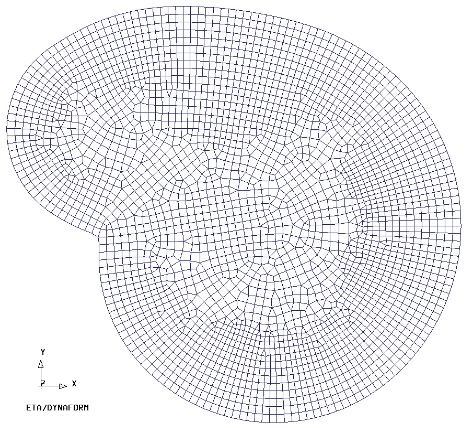


Fig. 12 Elements network of blank model created by usement of function “Part Mesh” and setting of element size 5

6.3 Comparison of blank models meshings

It is advantageous to comparise the elements networks of the blank model reciprocally. Complete view of elements which are situated in the mesh is got. Comparison of blank model meshings results is clearly shown in Tab. 1, where „Quad elements“ are elements of quad shape and „Triangle elements“ are elements of triangle shape.

Tab. 1 Comparison of blank model meshings results

Blank by TMSN	Set size of elements [-]	Number of elements [piece]	Quad elements		Triangle elements	
			[piece]	[%]	[piece]	[%]
Tool Mesh	10	647	570	88,1	77	11,9
	5	2425	2337	96,4	88	3,6
Part Mesh	10	753	723	96,0	30	4,0
	5	2907	2817	96,9	90	3,1

Quad elements are simpler for analysis and they dominate in both form of meshing. It is evident, while using meshing form „Tool Mesh“ the less elements will be in mesh than while using meshing form „Part Mesh“ (surface of blank is the same, size of elements is the same). It is because mesh of elements created by „Part Mesh“ will better conform to irregular shape-difficult blank. The meshing form „Tool Mesh“ creates elements network of regular elements in quad shape, but this elements network has not tendency to conform to irregular blank shape.

7 COMPARISON OF INTRICATE SHAPE STAMPING DEEP-DRAWING SIMULATION RESULTS WITH THE USE OF VARIOUS POSSIBILITIES OF BLANK MODEL MESHING

Simulation of drawing process of ventilator left cover stamping was carried out with the use of Dynaform 5.2 software at the Department of Mechanical Technology of Faculty of Mechanical Engineering of VŠB-Technical University of Ostrava on computer with operation system Windows XP.

In CAD program CATIA firstly model of stamping contained only lines and surfaces was created and stored like file iges. After import to Dynaform 5.2 software the network of elements was created (see 4) which is necessary for computing.

From die model a copy reduced about material thickness 0,9 mm and about technologic allowance 10 % was created in module „Preprocess“ by offset. So overall reduction of punch model was about 0,99 mm. At all tool parts models checking of the normals direction of separate model elements, which must have the same direction on all elements, checking of continuity of model boundary line and checking of double and overlap model elements was carried out.

Blank model was created in CAD program CATIA like file iges and additionally imported to Dynaform 5.2 software. At all models the same origin of coordinates was chosen, so after input of blank model to Dynaform 5.2 the blank must not be complicately centred and displaced.

For drawing process simulation all needed boundary conditions were defined like e. g. jako např. blank material, tool parts path, blankholder force, punch velocity, sliding friction coefficients.

For blank of ventilator left cover stamping a new material type was defined because material, from which the stamping is produced, in materials library of Dynaform 5.2 software did not exist. Material model of type 36 was used, which was created by Barlat and Lian in the year 1989 for modeling of sheet-metal at plane stress, to which the properties of material DC04 (11 305.21) evaluated by tensile tests were inputed: $R_{p0,2}$ = 205,5 MPa, R_m = 332,2 MPa, r_m = 1,76, n_m = 0,213.

Models of tool parts and blank were located to initial position, their properties were inputed, movement velocity, stroke and computed blankholder force corresponding to specific press 2 MPa. Computation was carried out in program LS-DYNA Jobs Submitter 2.2, which is part of Dynaform 5.2 software.

Simulation results can be seen in program ETA/Post-Proccessor 1.0, which is part of Dynaform 5.2 software. To program the resulting files d3plot or dynain can be inputed. File d3plot contain all informations about drawing simulation course, drawing process is here divided to some steps, which can be seen separately or to start the whole simulation like animation. The file dynain contains informations about the resulting drawing step, so it displays final stamping in the end of drawing simulation.

For comparison of intricate shape stamping deep-drawing simulation (the left cover of ventilator for truck Tatra 815) with the use of blank determined with the use of maximum shear stress trajectories (made from thin deep-drawing sheet-metal DC04) two kinds of analyses for all types of blank model meshing (see 5) were used – analysis of deformations and their visualization by forming limit diagram and analysis of stamping thickness (including points to assign the thickness).

7.1 Deep-drawing simulation results with the use of blank model meshed by function „Tool Mesh“ and setting of element size to 10

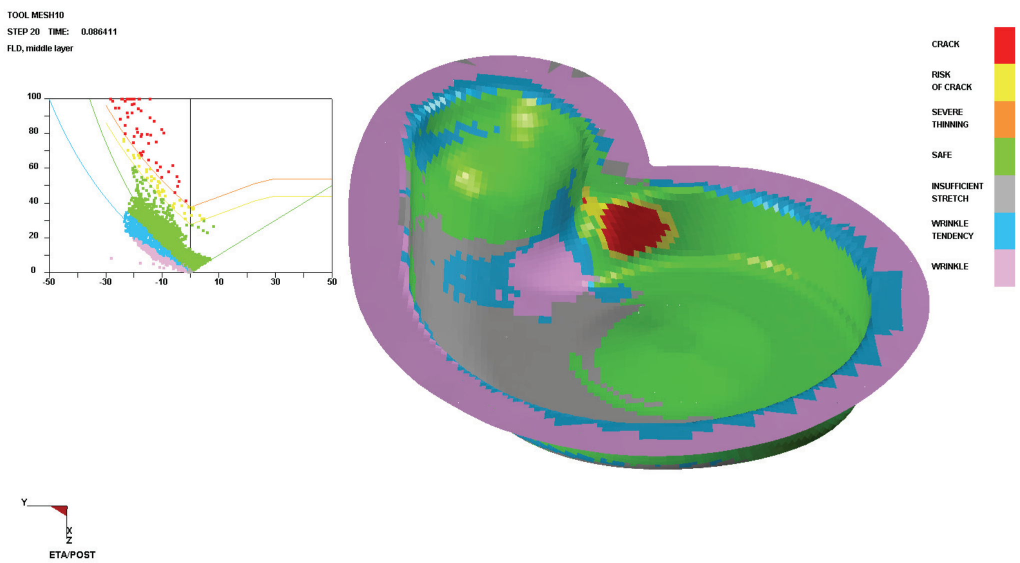


Fig. 13 Analysis of deformations and their visualization by forming limit diagram with the use of blank model meshed by function „Tool Mesh“ and setting of element size to 10

Elements on Fig. 13 are too large and they have brightly visible angles. The final mesh is very rough, that is why incomplete description of the situation on the stamping and so inaccurate simulation result exists.

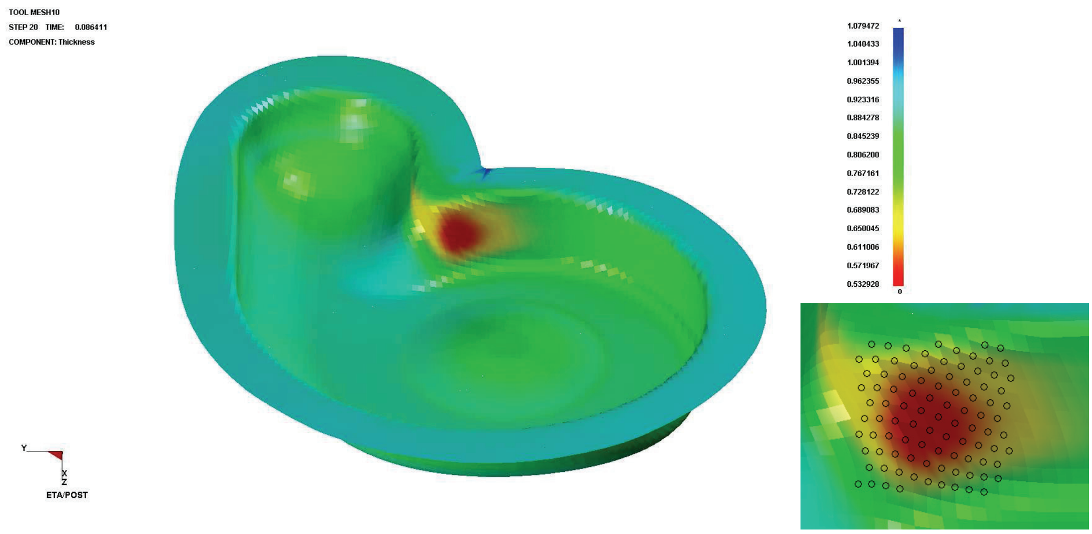


Fig. 14 Analysis of stamping thickness (including points to assign the thickness) with the use of blank model meshed by function „Tool Mesh" and setting of element size to 10

Determination of stamping final thickness values is carrying out by function „Control Option". On the small picture on the right (Fig. 14) is seen, that points for thickness determination are very long distance from each other and these points are very few. They insufficiently cover the critical place. This negative affects results – it is impossible to determine fairly exactly the minimum thickness, which is 0,53293 mm here.

7.2 Deep-drawing simulation results with the use of blank model meshed by function „Tool Mesh" and setting of element size to 5

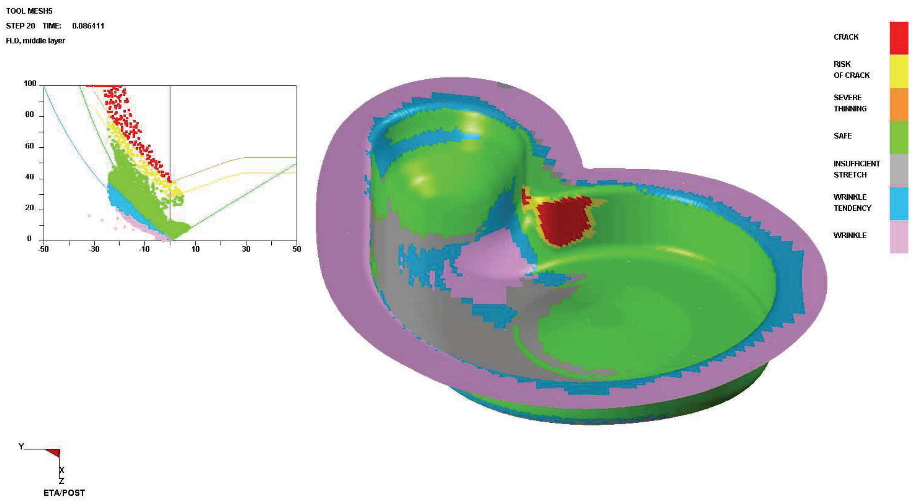


Fig. 15 Analysis of deformations and their visualization by forming limit diagram with the use of blank model meshed by function „Tool Mesh" and setting of element size to 5

The angles of elements are less visible (see Fig. 15), because the elements are smaller. The mesh is much soft, it better and more accurately describes situation on whole stamping and even in critical place. The user has better idea of possible real result, thanks to this much soft mesh.

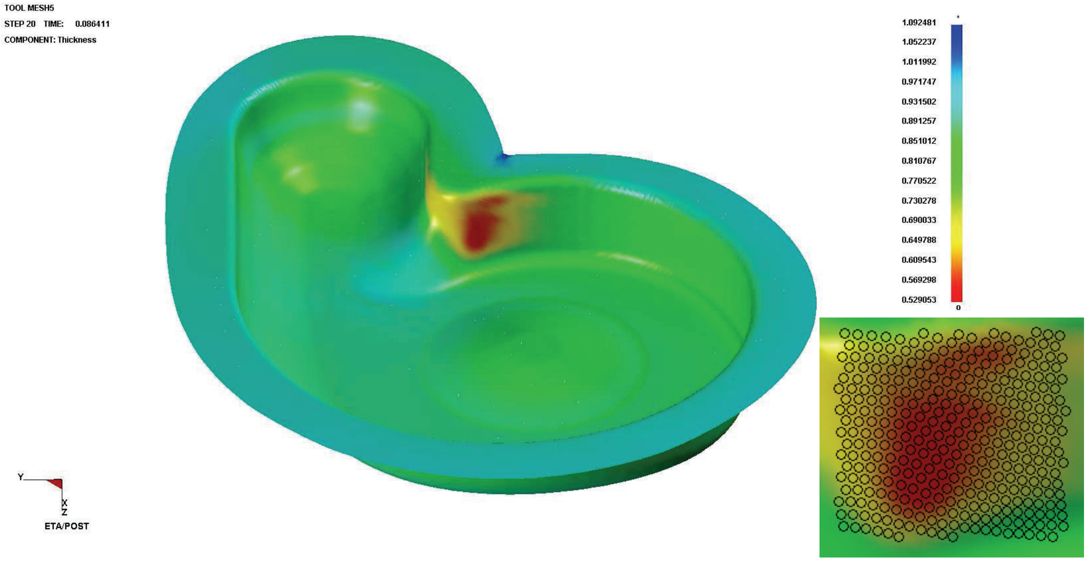


Fig. 16 Analysis of stamping thickness (including points to assign the thickness) with the use of blank model meshed by function „Tool Mesh“ and setting of element size to 5

The points for thickness determination are small (see Fig. 16), they are close together and more points exists on the controlled place. This is useful for determination of minimum thickness value. It means, that the situation will be analyzed better. The lowest value of thickness in critical place is 0,52905 mm, the second lowest value is 0,53191 mm. It is possible that this second lowest value was found like lowest value in form of meshing with element size 10.

7.3 Deep-drawing simulation results with the use of blank model meshed by function „Part Mesh“ and setting of element size to 10

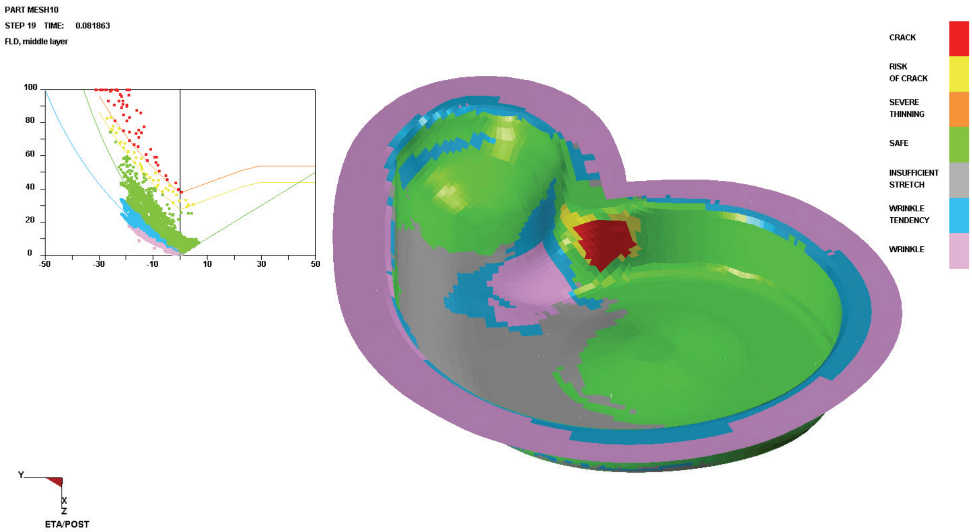


Fig. 17 Analysis of deformations and their visualization by forming limit diagram with the use of blank model meshed by function „Part Mesh“ and setting of element size to 10

The elements are very large (see Fig. 17), they do not make possible to analyse enough the situation on the stamping. The final mesh is very rough and so inaccurate for analysis. It is good to take a note, that possible crack has different shape than in „Tool Mesh“ form.

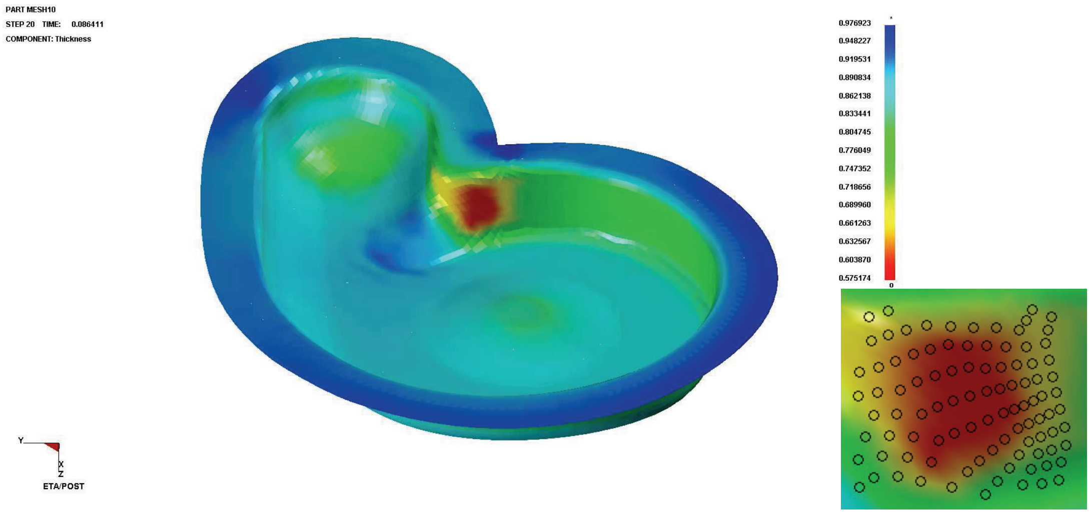


Fig. 18 Analysis of stamping thickness (including points to assign the thickness) with the use of blank model meshed by function „Part Mesh“ and setting of element size to 10

The points for thickness determination are very long distance from each other and these points are very few (see Fig. 18). They insufficiently cover the critical place, it may has negative affect for determination of minimum thickness value. It is impossible fairly exactly determine the minimum thickness. The lowest value of thickness is 0,57517 mm in critical place. This value incompletely describes the situation on the stamping.

7.4 Deep-drawing simulation results with the use of blank model meshed by function „Part Mesh“ and setting of element size to 5

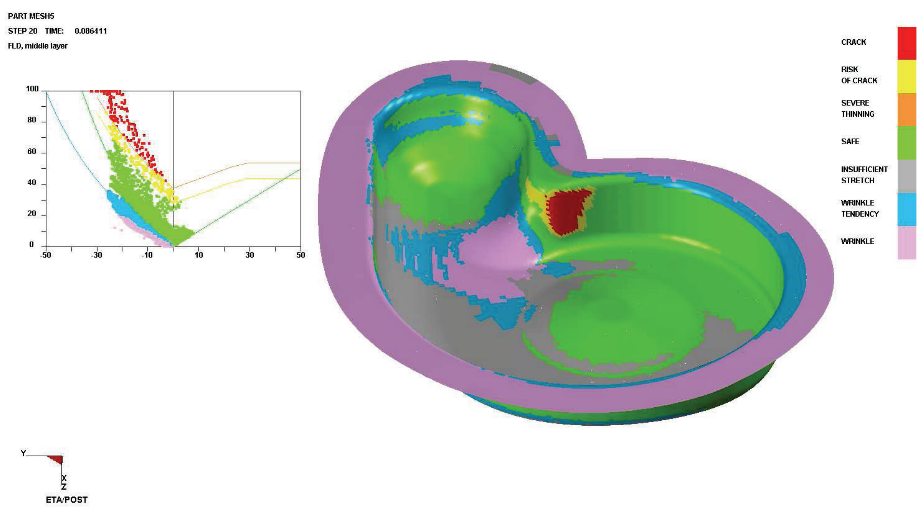


Fig. 19 Analysis of deformations and their visualization by forming limit diagram with the use of blank model meshed by function „Part Mesh“ and setting of element size to 5

The angles of elements are less visible, because the elements are smaller (see Fig. 19). Distribution of elements better describes flow of material while drawing, the elements are customized by way along direction of drawing (see Fig. 12), but in form „Tool Mesh“ the blank model is filled by straight quad elements no matter the incoming drawing (see Fig. 10). The crack has different shape in critical place than when using blank model meshed by function „Tool Mesh“, it is smaller. Shape of this crack is a little different than when using blank model meshed by function „Part Mesh“ with element size 10 too, because elements are markedly smaller, these elements describe the critical place better than large and rough elements.

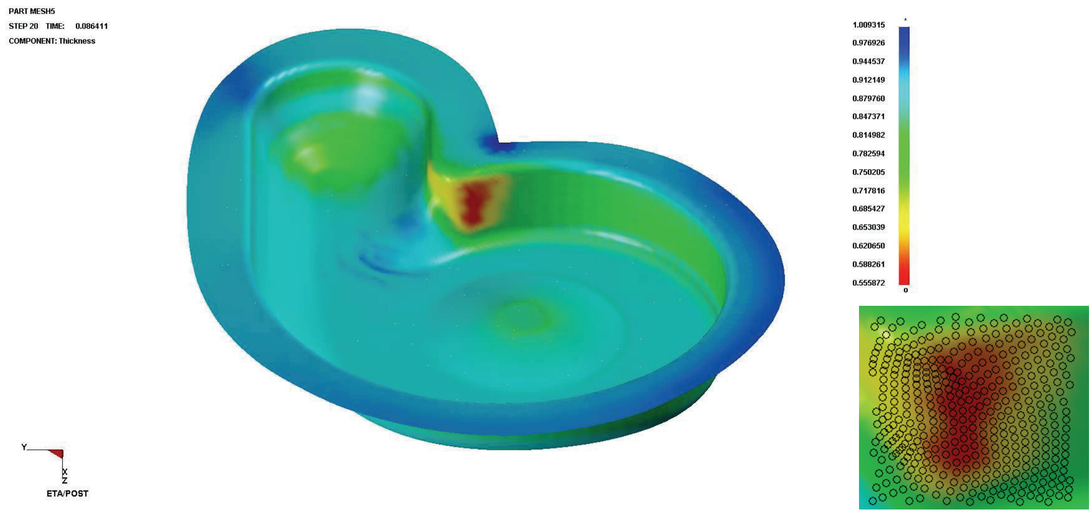


Fig. 20 Analysis of stamping thickness (including points to assign the thickness) with the use of blank model meshed by function „Part Mesh“ and setting of element size to 5

The points for thickness determination are small, they are very close together and these points are more on the controlled place (see Fig. 20). It makes the better description of critical place possible, for determination minimum value of thickness is it useful. The lowest value of thickness in critical place is 0,55587 mm, The value similar to the lowest value of thickness on stamping with element size 10 is the ninth lowest here.

8 CONCLUSIONS

Quality mesh for finite elements method consists of following, geometrically regular, approximately the same and regularly distributed elements on whole model surface.

All meshing methods noticed in the contribution were used on blank model determined by maximum shear stress trajectories method and on tool parts models for intricate stamping deep-drawing simulation – the left cover of ventilator for truck Tatra 815.

It was confirmed, that meshing form „Tool Mesh“ (see 4) is suitable for meshing of tool parts models (punch, die, blankholder) used for flat forming simulation. Before the final simulation it is necessary to find out and remove all mesh mistakes – it is necessary to check the normals direction of separate model elements, which must have the same direction on all elements, to check of continuity of model boundary line and to check of double and overlap model elements first of all.

From comparison of intricate shape stamping deep-drawing simulation results with the use of various possibilities of blank model meshing (see 7) was found out, that computing elements network on blank model created by meshing form “Part Mesh” (see 6) has thanks to elements shape and situation better conditions for subsequent drawing then elements network created by meshing form „Tool Mesh".

In case of problem of size of elements it was confirmed, that smaller elements created on the blank model are definitely more suitable for simulation of drawing (see 7) because the greater accuracy of an analysis achieves. It is surely possible to choose the smaller size of elements then which is mentioned in the contribution, but it is necessary to have needed powerful computer and longer analysis time must be supposed too. Computing time of finite elements method is influenced by procesor velocity, created network magnitude and by number of computed steps.

REFERENCES

[1] ČADA, R. *Tvářitelnost ocelových plechů : odborná knižní monografie.* Lektorovali: L. Pollák a P. Rumíšek. 1. vyd. Ostrava : REPRONIS, 2001. 346 s. ISBN 80-86122-77-8.

[2] ČADA, R. *Technologický design výtažků z plechu : teze inaugurační přednášky : Edice Habilitační a inaugurační spisy, sv. 39.* 1. vyd. Ostrava : Vysoká škola báňská - Technická univerzita Ostrava, Fakulta strojní, 2007. 68 s. ISBN 978-80-248-1459-9.

[3] FRODLOVÁ, B. *Řešení napjatosti a deformací tvarového výtažku z tenkého plechu : bakalářská práce.* Ostrava : VŠB-TUO, 2007. 91 s.

[4] ŠTEVKO, M. a ČADA, R. Zkoumání procesu tažení plechů s využitím metody konečných prvků. In *WORKSHOP 2006 Fakulty strojní.* 1. vyd. Red. R. Brázda a D. Frydrych. Ostrava : VŠB-TU Ostrava, 2006, s. 61-65. ISBN 80-248-0999-0.

[5] ČADA, R. a KIJONKA, M. Influence of input values changes on the simulation results in the Dynaform 5.2 software. In *Sborník vědeckých prací Vysoké školy báňské - Technické univerzity Ostrava : řada strojní.* Ostrava : VŠB-TU Ostrava, 2008, roč. 54, č. 1, s. 33-40. ISSN 1210-0471, ISBN 978-80-248-1891-7.

[6] ETA, Inc. *Eta/DYNAFORM User's Manual.* Version 5.2 Michigan : ETA, Inc., 2004. 360 s.

[7] ETA, Inc. *Eta/Post User's Manual*. Version 1.0 Michigan : ETA, Inc., 2004. 127 s.

[8] HRIVŇÁK, A. a EVIN, E. Simulácia procesov hlbokého ťahania. In *Mezinárodní vědecká konference Forming 99.* Red. Schindler, I., Radina, M., Kubina, T. Zlaté Hory : Katedra mechaniki i technoílogii przeróbki plastycznej, Politechnika Śląska Katowice, 1999, s. 176-181. ISBN 83-910722-1-5.