Weld Modeling with MSC.Nastran

J.Fang, C.Hoff, B.Holman, F.Mueller, D.Wallerstein
MSC.Software Corporation, Costa Mesa, Los Angeles, CA

ABSTRACT

A new general purpose connector element has been developed in MSC.Nastran. The element models the global load transfer in spot weld and seam weld connections. The weld element has a physical stiffness. Constraint equations are generated internally. The resulting element stiffness is always of proper rank. The user does not have to take precautions for in plane rotations in shells. Three types of connections can be defined with the new weld element, point to point, point to patch, and patch to patch. The connected elements can be of different type. The meshes do not have to be congruent. The user can request that the weld element is generated without explicit constraints, thereby avoiding the costly elimination of m-set degrees of freedom and increasing the performance for large models.

The connectivity and property of the new element is described. A short overview of the underlying constraint equations is given. Numerical experiments are conducted to show the accuracy, limitations, and overall performance of the new element. MSC's Automotive Modeling System for Full Vehicle Analysis MSC.AMS-FVA is used to show the spot weld modeling process.

INTRODUCTION

For a number of years, the automotive customers of MSC.Nastran have asked MSC for better tools to model spot welds. MSC has hesitated to develop a new special purpose modeling tool mainly because MSC.Nastran has a variety of capabilities to model connections, for example, flexible springs and bars (CBUSH, CBAR, etc.[1]), rigid elements (RBAR, RBE2, RBE3, etc. [1]) and multi-point constraints (MPC [1]). In recent years, criticism about the existing modeling tools has grown with the rapidly changing requirements of today's CAE process, see for example [2]. The meshes in automotive finite element models are getting finer to achieve higher accuracy. Users are constantly searching for ways to cut processing time. Modeling tools must be easy to use, must be generated fast, must fit into the design process, and must cope with demands in overall performance.

With the traditional tools of MSC.Nastran, spot welds are modeled using a point to point connection. The shell vertex grids of an upper and lower shell are connected with a flexible bar or a rigid element, see the bulk data entries CBAR and RBE2 [1]. A lot of automotive models exist using the point to point technique mainly because MSC did not offer more convenient tools. There are three disadvantages using the point to point connection.

1. Meshes of different parts have to match at the spot weld connection points. The two grids of the spot weld have to line up. Some parts must be re-meshed, most often with time consuming manual procedures.

2. The point to point connection underestimates the stiffness of the spot weld connection for large spot weld diameters. With finer meshes, the diameter of a spot weld is of the same order of magnitude or just one order of magnitude less than the element length. The diameter must be taken into account to simulate the force transfer from the spot weld to the shell correctly.

3. The point to point connection causes numerical problems in the finite element shell model. In Mindlin shell theory, the stiffness of the rotations normal to the shell plane does not exist. MSC.Nastran's QUAD4, for example, is a Mindlin shell element. In the point to point connection, rotations normal to the shell plane may be excited. No problems occur if the shell vertex grids of the upper and lower shell element are lined up so that the vector connecting the two vertex grid points in the direction normal to both the upper and lower shell plane. If the two shell vertex grids are not lined up, spurious modes may occur and the force transfer becomes inaccurate. Users have tried to defeat the pitfalls of this technique by using the parameter entry K6ROT [1] which puts a small penalty stiffness on the rotational degree of freedom of the shell.

The first and second disadvantage are inherent to the point to point connection and can only be overcome with other connection types. The third disadvantage can be removed by defining discrete normals at the shell vertex grids with the bulk data entry SNORM, see [1]. The two discrete normals of the upper and lower shell vertex grid must be in the direction of the vector connecting the two vertex grids. Several MSC.Nastran users have written pre-processing routines to generate the normals. Because a lot of models exist with point to point connections for spot welds, MSC has decided to offer a tool to overcome the third disadvantage by generating shell normals automatically, see below.

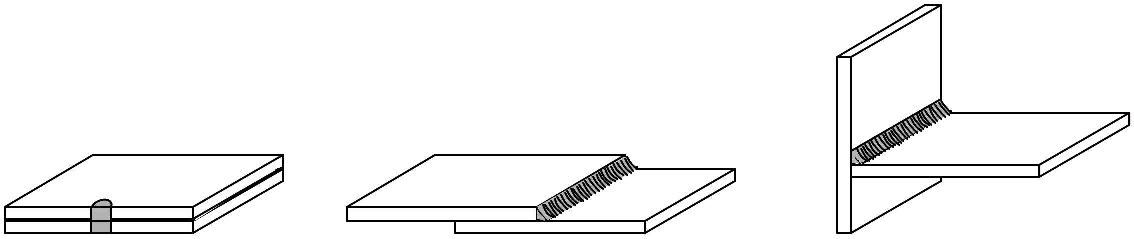
Most efforts in spot weld modeling are now concentrating on two requirements. First, the spot weld modeling tool must be able to connect non congruent meshes so that parts do not have to be re-meshed. Second, the area of the spot weld must be taken into account to get a correct force transfer from the weld into the shell.

In recent years, spot weld generators have been developed by several third party software vendors using existing finite element tools and generating input for solvers like MSC.Nastran. All these products start with an existing finite element model without spot welds. The information of the spot weld locations is added from a separate file containing a list of spot weld grid points with xyz locations. The spot weld information comes from a CAD model or a special spot weld modeling system. The projection of the spot weld grid to two or more shell elements is found. Then, the spot weld itself and its connection to the shell elements is generated. For example, in [3], the spot weld itself is modeled with a HEXA element which has the actual size of the weld nugget. The HEXA grids are connected to the three translational degrees of freedom of the shell vertex grids with RBE3 type elements. This modeling technique fulfills the two main requirements, non congruent meshes and area consideration. MSC's Automotive Modeling System - Full Vehicle Analysis MSC.AMS-FVA [4] offers an integrated spot weld modeling package including MSC.Nastran and the spot weld generator described in [3]. The package is used by several MSC clients in the automotive industry.

Although the solution in [3] and other products on the market fulfill all requirements for spot weld modeling today, MSC has decided to offer an in-house solution with a general purpose connector element in MSC.Nastran. MSC wants to be able to offer a complete package including graphical user interface, spot weld generator, and solver.

The German Volkswagen AG has approached MSC about two years ago and has proposed an input definition of a spot weld in MSC.Nastran which is close to the definition used in PAMCRASH [5] . MSC has written a proto-type which handles non congruent meshes. The results have been presented a year ago [6].

After a successful launch of the proto-type, MSC has improved and extended the weld element. The area of the weld is taken into account. In addition, the weld element has been extended to a general purpose connector element which can model other types of connections and can connect arbitrary element types. Besides spot welds, the element can be used to model seam welds, see Figure 1. The element can also be used to model bolts, screws, and engine mounts. In this paper, we will only discuss the spot weld connection.



Spot **B-Seam** **T-Seam**

Figure 1. Weld

We start with a description of the connectivity and property of the new element. The underlying constraint equations are explained. Then we show some numerical experiments to demonstrate the accuracy and the limitations of the new element. Finally, we show MSC.Nastran's spot weld capability integrated into MSC.AMS-FVA [4].

SPOT WELD CONNECTIVITY AND PROPERTY DEFINITION

With the new CWELD element, the user can define three types of connections.

1. For a point to point connection, an upper and lower shell vertex grid, GA and GB, are connected, see Figure 2. The weld axis is the vector from GA to GB. Two shell normals are automatically generated for the upper and lower shell vertex grid, both normals point in the direction of the weld axis. The user does not have to specify PARAM, SNORM to activate the generation of normals.

2. For a point to patch connection, a vertex grid point GS of a shell element is connected to a surface patch, see Figure 3. The patch is either defined by grid points GAI or a shell element SHIDA. A shell normal in the direction normal to the patch is automatically generated and put on GS.

3. For a patch to patch connection, a spot weld grid GS is connected to an upper and lower surface patch. GS may or may not lie on the patches, see Figure 4. The upper patch is defined by grid points GAI or by a shell element SHIDA, the lower patch by GBI or SHIDB.

The second and third type are defining the most general connectivity. In addition, the third type meets the requirements of non congruent meshes and weld area consideration. The spot weld grids GS are usually not associated to a finite element mesh, they are generated in a CAD system or a special purpose automotive modeling system like MSC.AMS-FVA [4]. The spot weld grid GS may or may not lie on the finite element geometry. The weld element geometry is determined by projecting the spot weld grid GS on the finite element faces of the upper and lower shell element. First, GS is projected normal to the upper shell, the piercing point is called grid GA. Along the direction GS-GA, the piercing point GB on the lower shell is determined. The projected grids, GA on the upper shell and GB on the lower shell, are defining the element length and direction of the spot weld itself, see Figure 4.

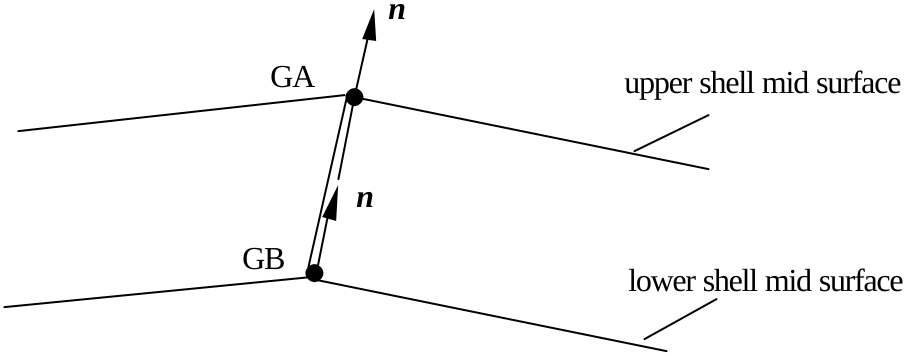


Figure 2. Spot Weld with Point to Point Connection.

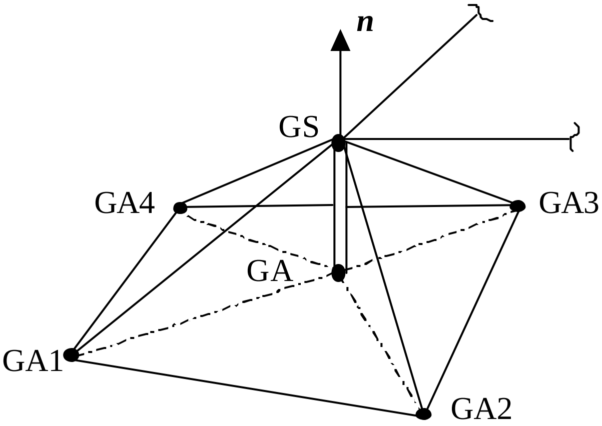


Figure 3. Spot Weld with Point to Patch Connection.

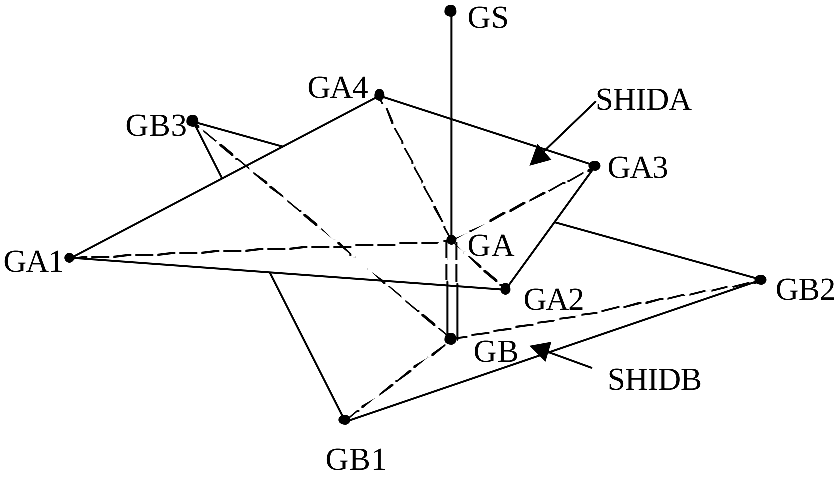


Figure 4. Spot Weld with Patch to Patch Connection.

If more than two layers need to be connected, a second CWELD element is defined which refers to the same spot weld grid GS as the first CWELD. The projection vector of the first CWELD determines the weld element axis and is maintained for all following layers. Up to 10 layers can be connected. Elements of different types can be connected, for example, a CQUAD8 and a CTRIA3. The spot weld grid GS can be at an arbitrary location as long as the projected grids GA and GB lie in the upper and lower surface patch, respectively. After GA and GB are generated, the CWELD element connectivity is stored in a generalized form, the grids GS, GA, GB, grids GAI of the upper surface patch and grids GBI of the lower surface patch. A surface patch must have at least 3 grids and can not exceed 8 grids. A surface patch does not have to correspond to an element.

The property of a CWELD element is defined on a PWELD entry. The property parameters are the material identification number and the diameter D of the spot weld. The property entry has additional parameters to describe seam welds. In addition, the user can define flags for advanced features. For example, the user can ask to include all constraint equations of the element in the stiffness matrix and thereby avoid the creation of m-set degrees of freedom. Without explicit m-set degrees of freedom, the resulting system of equations is more robust and faster to solve.

FINITE ELEMENT REPRESENTATION OF THE CONNECTOR

The weld itself is modeled with a special shear flexible beam type element with length L and a circular cross section with diameter D. The length L is the distance of GA to GB. The element has 2x6 degrees of freedom. The element has been developed to behave well for very small ratios of length L to diameter D, see Figure 5. The range of the ratio L /D is restricted to $0.2 \le L / D \le 5.0$ by default. The program overwrites the length L if it is outside the range to protect against spot welds with zero length and overly long spot welds. The user can change the range on the property entry.

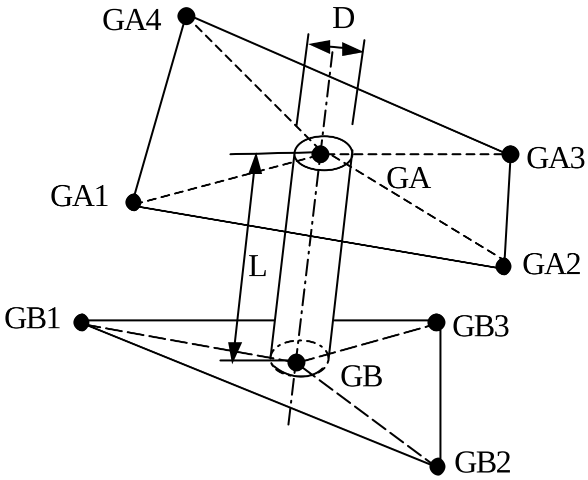


Figure 5. Spot Weld Element

For the point to point connection, two normals are generated at grid GA and GB in the direction of the weld axis. For the point to patch and patch to patch connection, the degrees of fredom of the spot weld end points GA and GB are constrained as follows. The translational and rotational degrees of freedom of spot weld end grid GA are connected to the translational degrees of freedom of grids GAI with constraints from Kirchhoff shell theory,

$$\begin{Bmatrix} u \\ v \\ w \end{Bmatrix}_A = \sum N_I(x_A, h_A) \cdot \begin{Bmatrix} u \\ v \\ w \end{Bmatrix}_I \tag{1}$$

$$\begin{aligned} q_x^A &= \frac{\partial w}{\partial y} = \sum N_{I,y} \cdot w_I \\ q_y^A &= -\frac{\partial w}{\partial x} = -\sum N_{I,x} \cdot w_I \\ q_z^A &= \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right) = \frac{1}{2}\left(\sum N_{I,x} \cdot v_I - \sum N_{I,y} u_I \right) \end{aligned} \tag{2}$$

The 6 equations are written in the local tangent system of the surface patch at point GA. The two tangent directions are x and y, the normal direction is z. N_I are the shape functions of the surface patch, x_A and h_A are the normalized coordinates of GA, u, v, w are the displacements, q_x, q_y, q_z are the rotations in the local tangent system at GA. For grid GB, another set of 6 equations similar to (1) and (2) are written resulting in 12 constraint equations.

In summary, the spot weld element consists of a two node element with 12 degrees of freedom and 12 constraint equations. The user has two options to process the constraint equations. In the first option, the constraints are generated as explicit multi-point constraints, the 2x6 degrees of freedom of GA and GB are put into the dependent set (m-set). In the second option, the 2x6 constraint equations are worked into the 12x12 stiffness matrix of the spot weld element. The resulting element is a 3xN degrees of freedom element where N is the total number of grids GAI plus GBI. The second option does not generate m-set degrees of freedom in the problem. The costly m-set elimination is avoided. Furthermore, occasional problems with singular constraint matrices are avoided.

EXAMPLES

SPOT WELDS MODELED WITH THE POINT TO POINT CONNECTION

We investigate a simple model of a two layered plate to demonstrate the numerical problems with traditional modeling techniques of point to point connections. Two sheets of 1x30x90 mm are connected with spot welds, see Figure 6. Each sheet is modeled with a mesh of 3x9 CQUAD4 elements, the meshes are connected with two rows of 8 spot welds using the point to point connection. The spot welds are modeled in two ways.

1. The traditional modeling technique using rigid bars (RBARs) and K6ROT.
2. The new CWELD element with the point to point connection.

We compare the results of three different meshes of the upper layer. First, the upper mesh is perfectly aligned with the lower mesh. This mesh is the baseline. For the second and third mesh, the inner grids of the upper layer are slightly perturbed with an amplitude of 0.2 mm in the y-direction. The second mesh has a wavy pattern, the third mesh has a uniform shift.

2 x (3 x 9) Quad4s	t = 1.0 mm	E = 2.06 e+5 N/ mm^2
2 x 8 Spot Welds	D = 2.0 mm	ν = 0.3
Perturbation	Δ y = 0.2 mm	r = 0.785 e - 8 kg/mm ^{2}

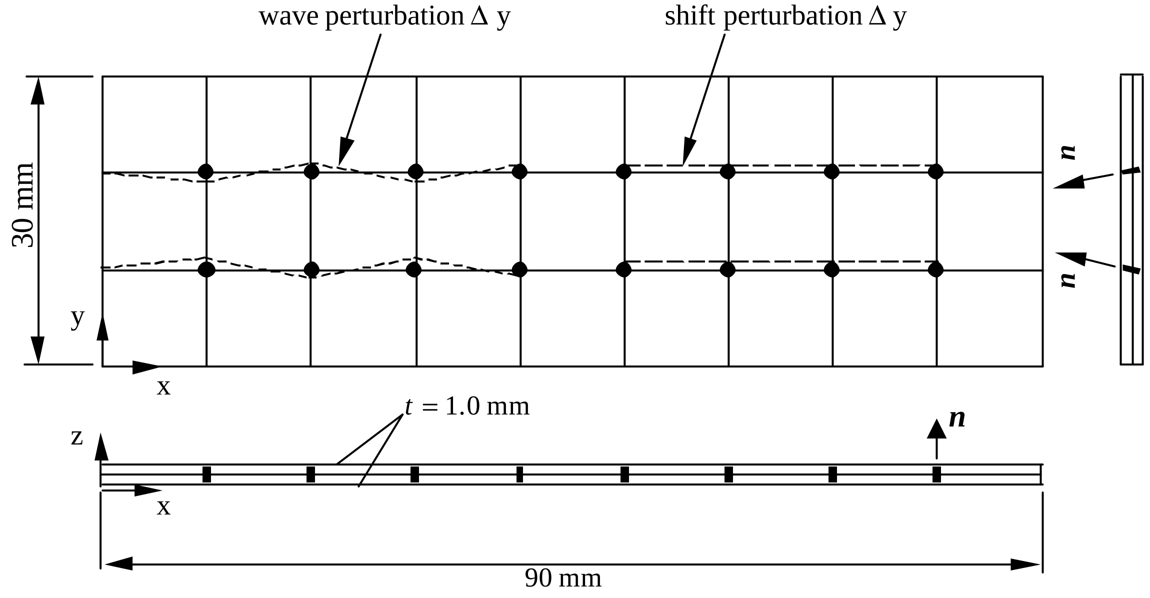


Figure 6. Two Plates Connected with 16 Spot Welds

The eigenfrequencies of the model are calculated in MSC.Nastran Sol 103. For the perfectly aligned mesh, the results of the traditional technique using RBARs and K6ROT are nearly identical to the new CWELD element. For the two perturbed meshes, we do not expect a significant change in the results because the outer grids are still perfectly aligned. With the traditional modeling technique the eigenfrequencies are underestimated by 6 to 12 %, see Figure 7. With the CWELD element, the results of the two perturbed meshes are nearly identical to the perfectly aligned mesh, the errors are so small that they are not visible in Figure 7.

If K6ROT is not used in the traditional modeling technique with RBARs, then the error is much larger and spurious modes may occur. With the new CWELD element K6ROT is not necessary.

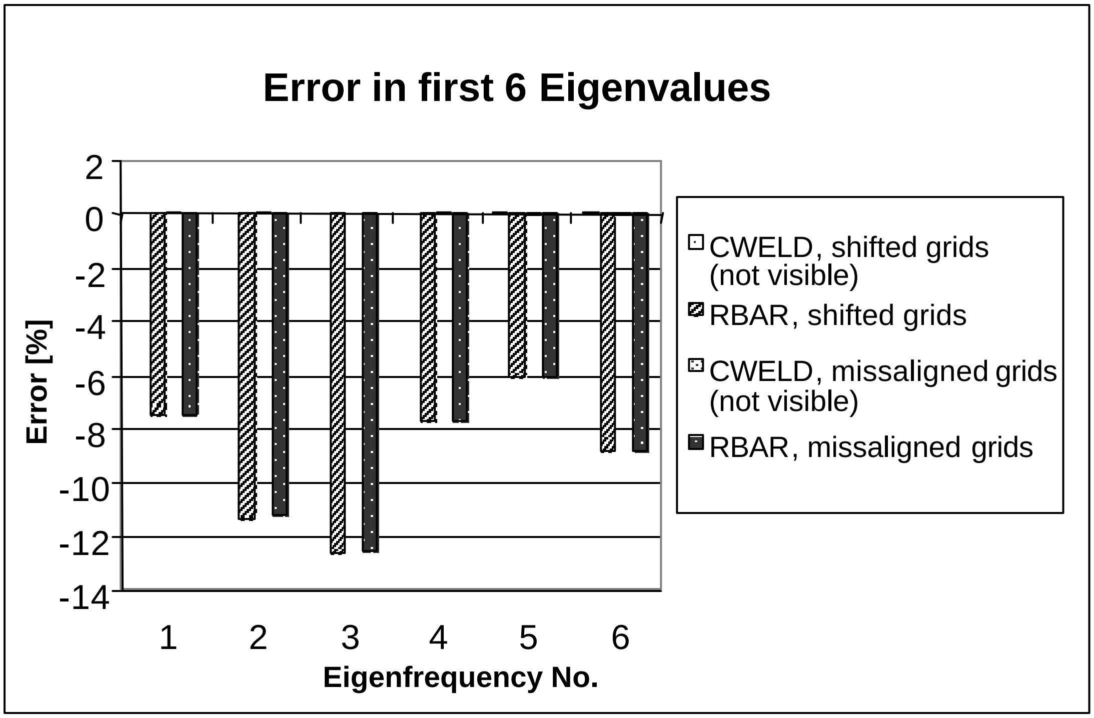


Figure 7. Errors in the first 6 Eigenfrequencies

A large scale model of a body in white (BIW) with point to point connection of spot welds has been converted from the traditional modeling technique with RBE2s and K6ROT to the new CWELD elements. For the first 40 eigenfrequencies, the RBE2 and K6ROT model is about 1.0 to 2.5 % softer than the CWELD model.

SPOT WELDS MODELED WITH THE PATCH TO PATCH CONNECTION

The stiffness of the patch to patch connection is investigated with a simple model of two QUAD4 elements connected with one CWELD element. The diameter D of the spot weld is varied from

$$0.01 \leq \frac{D}{S} \leq 1.0$$

where S is the element length. We look at deformations for the following loads and boundary conditions: in plane shear, in plane twist, transverse shear, and pure bending, see Figures 8 to 11. We compare the displacements of the extreme case D/S=1.0 with two other models. First, we use a HEXA element to connect the two QUAD4 elements. The HEXA element has the same dimensions as the QUAD4 elements in the x-y plane and a height equal to the thickness of one shell in the z-direction. Second, we use one shell element with double thickness.

All displacements are normalized with the displacements of the HEXA model with D/S=1.0. The normalized displacements are plotted against the ratio D/S in a double logarithmic scale to show the dependency of the displacements with respect to the diameter D. The plots show a clear dependency of the deformations with respect to the spot weld diameter that is consistent with continuum theory. For the extreme case of D/S= 1.0, the CWELD is softer than the CHEXA in shear and twist. However, compared to the shell with double thickness, the CWELD is stiffer in shear and twist. Overall, the new CWELD element has a reasonable stiffness for diameters which do not exceed the size of an element.

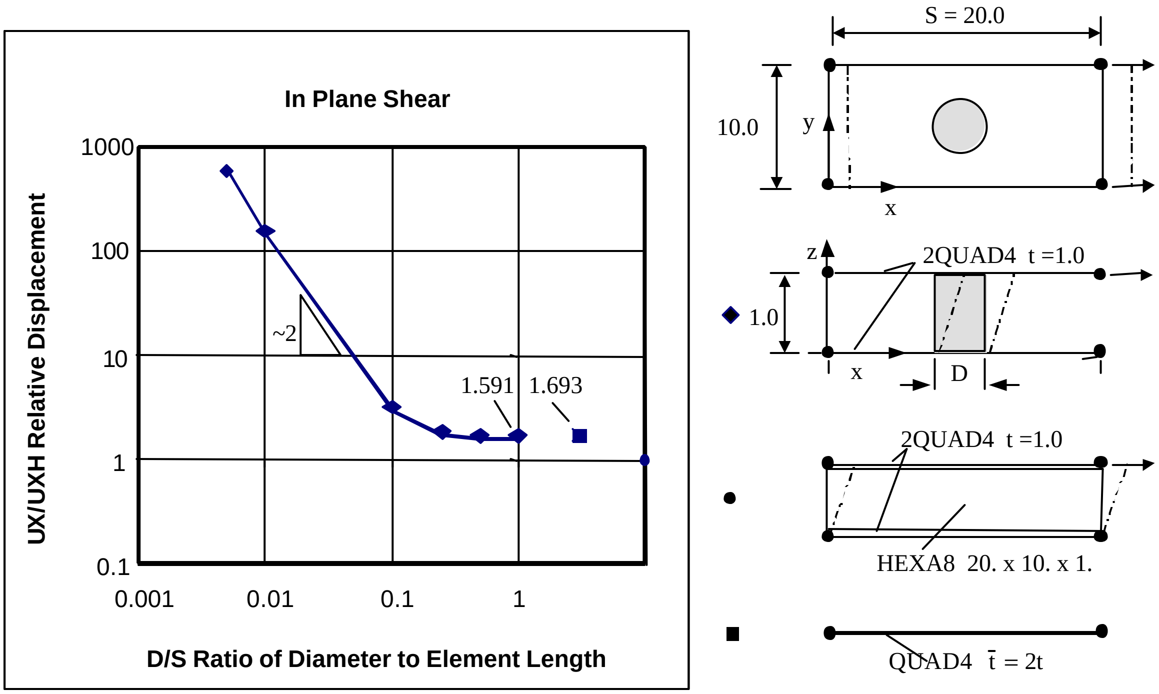


Figure 8. Spot weld deformation for various spot weld diameters, in plane shear

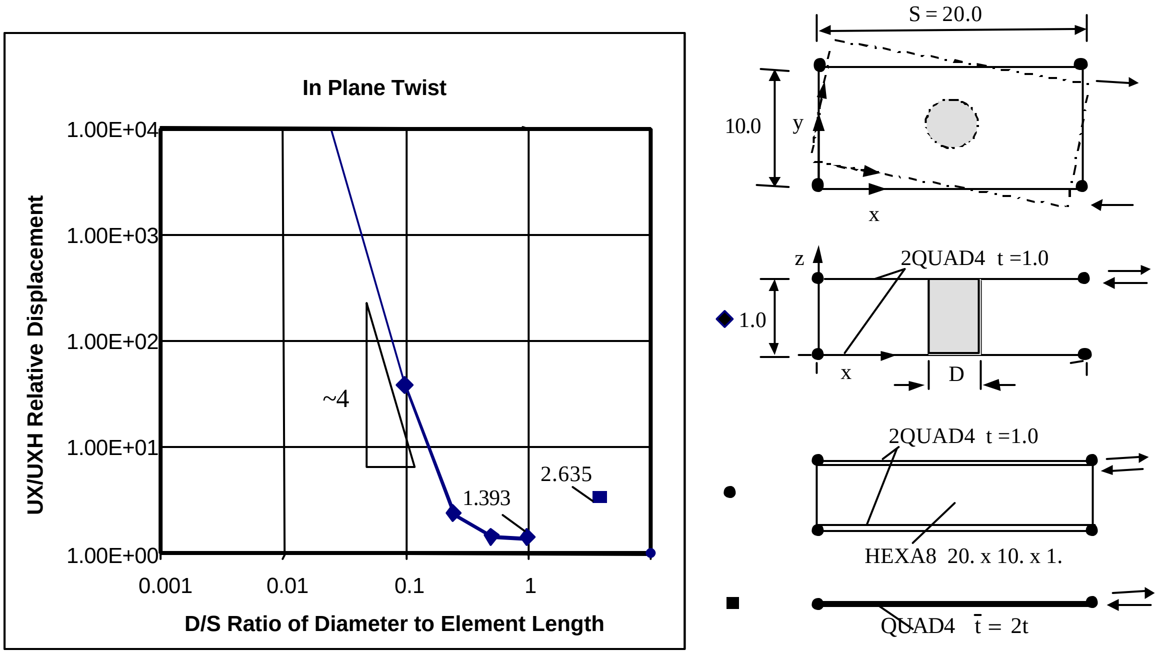


Figure 9. Spot weld deformation for various spot weld diameters, in plane twist.

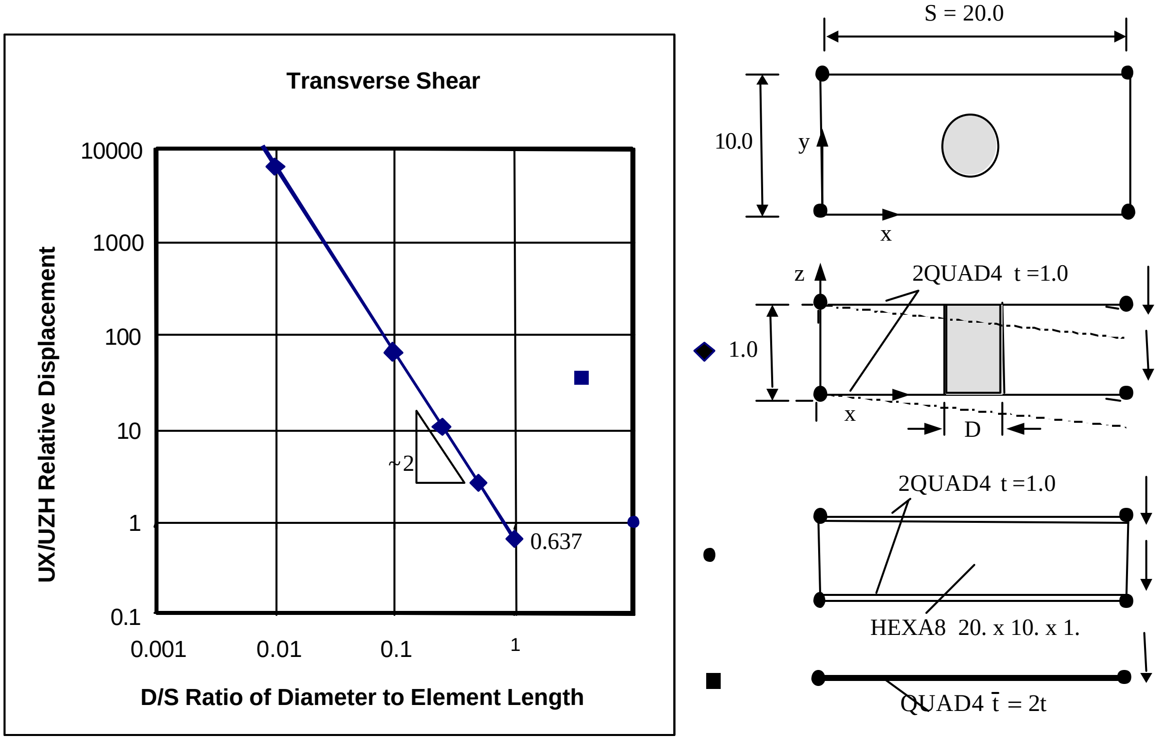


Figure 10. Spot weld deformation for various spot weld diameters, transverse shear.

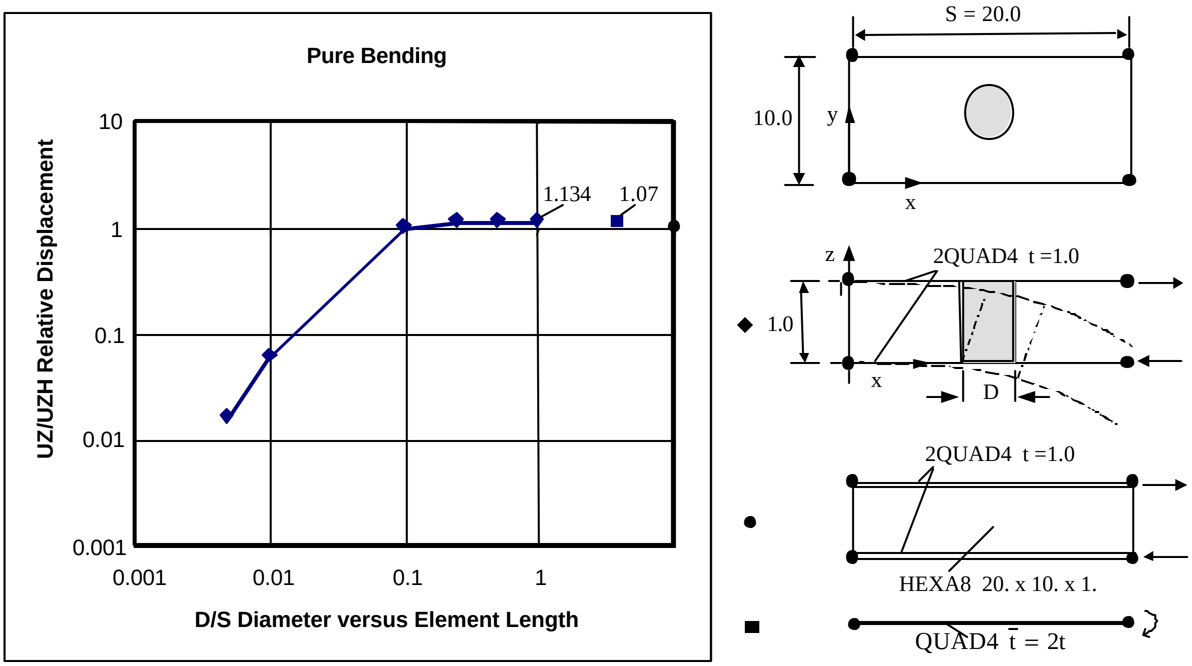


Figure 11. Spot weld deformation for various spot weld diameters, pure bending

SPOT WELD MODELING PROCESS WITH MSC.AMS-FVA AND MSC.NASTRAN

Two automotive parts are connected with spot welds. The parts have non congruent meshes. The finite element model of both parts exist in form of a MSC.Nastran input file. In MSC.AMS-FVA the user specifies the spot weld locations GS or reads the location from a file generated by a CAD system. MSC.AMS-FVA finds the connection of the spot weld grid GS to the upper and lower shell element, generates CWELD elements and merges them into the existing MSC.Nastran input file. The model of the connected parts is then analyzed in MSC.Nastran. Several Body in White FE models from automotive customers have been processed successfully with MSC.AMS-FVA and MSC.Nastran. The results of the new CWELD match experiments within sufficient accuracy and furthermore, agree with results from other simulation techniques, for example with CDH-Spot [3].

CONCLUSIONS

MSC has developed a general purpose connector element which can connect two vertex shell points, a vertex shell point and a surface patch or two surface patches. The element fulfills the two basic requirements for spot welds, it can join non coincident meshes and it takes the area of the spot weld into account. The new spot weld element simulates the force transfer between two surface patches accurately for $D/S \leq 1.0$ where D is the spot weld diameter and S is the mesh size. If the diameter is larger than the surface patch, the spot weld element may underestimate the stiffness of the connection. The weld element is always of proper rank. For high performance analysis, the user has the option to avoid the generation of explicit constraints. The accuracy and efficiency of the new element has been proven in the analysis of several large automotive models.

OUTLOOK

Fatigue analysis of spot welds requires high quality stresses. However, with the modeling technique described here, the stresses in the shell elements around the spot weld grid points GA and GB are not accurate enough. There are two ways to get more accurate stresses.

1. A new detailed model around the spot weld is created. A couple of elements of the coarse model around the spot weld are cut out, a finer mesh is created, grid point forces and boundary conditions of the coarse model are transferred to the fine model. Then a stress analysis is done on the fine model, see for example [7].
2. For a quick estimate of the maximum stress in the shell, stress intensity factors have been developed, see for example [8]. The stress values from the finite element patch are multiplied by stress intensity factors to estimate stress peaks around the weld nugget.

Fatigue is just one of many opportunities to develop more tools for the spot weld modeling process.

REFERENCES

[1] *MSC.Nastran Quick Reference Guide, Version 70.5*, MSC.Software, Costa Mesa, Los Angeles, CA, 1998.

[2] A.Jonscher, M.Lewerenz, C.Hoff, "MSC.Nastran's New Spot Weld Element in the CAE Process", Proceedings of the 2nd MSC Worldwide Automotive User Conference, Dearborn, MI, 2000.

[3] D.Heiserer, M.Chargin, J.Sielaff, "High Performance, Process Oriented Weld Spot Approach",1st MSC Worldwide Automotive User Conference, Munich, Germany, 1999.

[4] *MSC.AMS- FVA* Automotive Modeling System, Full Vehicle Analysis, MSC.Software, Costa Mesa, Los Angeles, CA, 1999.

[5] *PAMCRASH Reference Manual, PAM-SCL Version 1998*, ESI/PSI Wiesbaden, Germany.

[6] A.Jonscher, M.Lewerenz, G.Luehrs, "The New Spot Weld Element in the CAE Process", Proceedings of the 1st MSC Worldwide Automotive User Conference, Munich, Germany, 1999.

[7] X.Deng, W.Chen, G.Shi, "Three-Dimensional Finite Element Analysis of the Mechanical Behavior of Spot Welds," *Finite Elements in Analysis and Design*, Vol.35, 2000, pp.17-39.

[8] Shicheng Zhang, "Recovery of Notch Stress and Stress Intensity Factors in Finite Element Modeling of Spot Welds", Proceedings of NAFEMS World Congress 99, Newport, RI, USA, April 1999.