MSC.Nastran's New Spot Weld Element in the CAE-Process

A. Jonscher, M. Lewerenz, G. Lührs,
Volkskwagen A.G., Wolfsburg, Germany

ABSTRACT

The Volkswagen AG and MSC.Software Corporation have developed a new spot weld element in MSC.Nastran. The new element allows an automatic assembly process for large scale finite element models of bodies in white.

Starting point was the spot weld philosophy of Volkswagen, which is based on a one point connection for each spot weld. Due to the need of congruent meshes of the flanges, it requires a large time for the model generation.

The new element follows the idea of a connection between shell elements at any point within the elements, not only at element nodes. Therefore, there is no need for congruency between the part models and each single part can be modeled independently. Furthermore, the part assembly can be done automatically by using the spot weld information from CAD systems.

In this paper, we describe the traditional and the new spot weld modeling process. We demonstrate the capability and accuracy of MSC.Nastran's new spot weld element on models of body parts and a body in white. We compare results with experiments. Compared to the traditional spot weld modeling process, we show significant simplifications and time savings when the new spot weld element is used.

INTRODUCTION

General discussion of CAE process.

…it is easy to see that the generation of congruent meshed flanges takes a large part of it, see figure 4. Besides the effort of re-meshing, the point wise connection has other disadvantages which are discussed in [2]. About two years ago, VW has approached MSC with the idea of a mesh independent spot weld element, ...

EXAMPLES

PRINCIPAL EXAMINATIONS ON A HAT PROFILE
BONNET STIFFNESS
BODY IN WHITE

CONCLUSIONS

Dicussion about process time savings.

OUTLOOK

APPENDIX

SPOT WELD CONNECTIVITY AND PROPERTY DEFINITION

The usual spot weld information from a CAD system is a location in space, we call the location the spot weld grid GS. In addition, a diameter D may be specified. After a CAD model is transformed into a finite element model, the spot weld grid GS may or may not lie on the finite element geometry. The spot weld geometry in the finite element model 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.

In MSC.Nastran, it is assumed that a pre-processor provides the connectivity of the spot weld grid GS to two or more shell elements or element faces. Two connectivity entries CWELD define the connection of GS to an upper and lower shell element, respectively, see Figure A1. Elements of different types can be connected, for example, a CQUAD8 can be connected to 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 element, respectively. After GA and GB is generated, the CWELDG element connectivity is stored in a generalized form, the grids GS, GA, GB, upper element face grids GAI and lower element face grids GBI. A face must have at least 3 grids and can not exceed 8 grids.

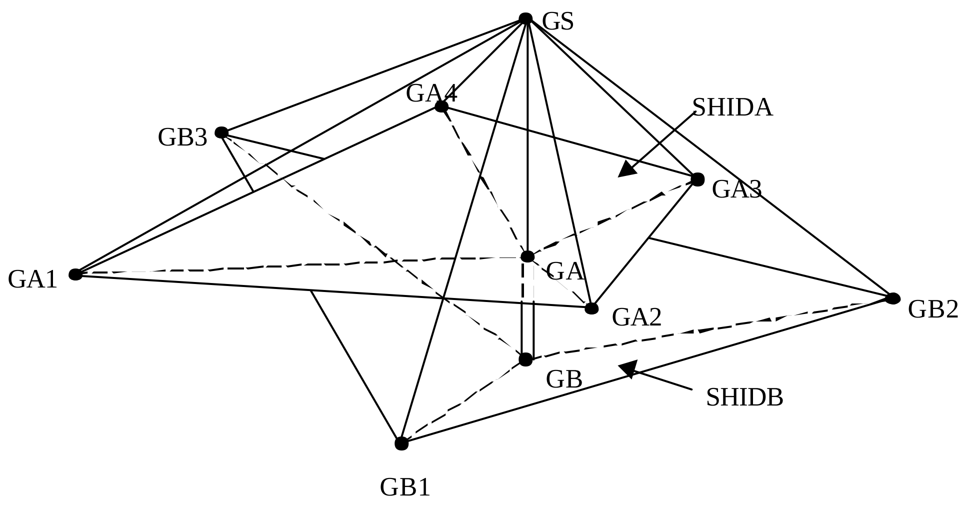

Figure A1. CWELD Connectivity

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. In addition, the user can define flags for advanced features, see [2].

The connectivity definition described above is the most general definition to join non congruent meshes of any element type with spot welds. The program allows other connectivity definitions which are not discussed here, because they are not as general as the option described above, see [2] for further discussions of other options. The most important feature of the new spot weld connectivity entry CWELD is that the user does not have to remesh the parts when connecting them.

FINITE ELEMENT REPRESENTATION OF THE WELD

The spot weld element itself is modeled with a special shear flexible beam type element. The element has a circular cross section with diameter D and the length 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. The recommended range of the ratio length L to diameter D is

$$0.2 \leq \frac{L}{D} \leq 5.0$$

The program overwrites the length L if it is outside the range. The user can change the range on the PWELD entry.

The translational and rotational degrees of freedom of spot weld end grid GA are connected to the translational degrees of freedom of shell grids GAI with constraints from Kirchhoff shell theory,

$$\begin{Bmatrix} u \\ v \\ w \end{Bmatrix}_A = \sum N_I(x_A, h_A) \cdot \begin{Bmatrix} u \\ v \\ w \end{Bmatrix}_I \qquad \text{(A1)}$$

$$q_x^A = \frac{\partial w}{\partial y} = \sum N_{I,y} \cdot w_I$$

$$q_y^A = -\frac{\partial w}{\partial x} = -\sum N_{I,x} \cdot w_I \qquad \text{(A2)}$$

$$q_z^A = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right) = \frac{1}{2}\left(\sum N_{I,x} \cdot v_I - \sum N_{I,y} u_I\right)$$

The 6 equations are written in the local tangent system of the shell 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 described by grid points GAI. Xeta_A and eta_A are the normalized coordinates of GA. U,v,w are the components of the translations, theta_x, theta_y, and theta_z are the componenets of the rotations. For grid GB, another set of 6 equations similar to (1) and (2) are written resulting in 12 multipoint constraint equations. Parameter studies show that the constraint equations give accurate results in the range of

$$0.05 \leq \frac{D}{S} \leq 1.0$$

where S is the characteristic length of the shell element.

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 multipoint 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 and GBI. The second option avoids the generation of m-set degrees of freedom in the problem. The costly m-set elimination is avoided. Furthermore, occasional problems with singular constraint matrices are avoided.

REFERENCES

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

[2] J.Fang, C.Hoff, B.Holman, F.Mueller, D.Wallerstein , “Weld Modeling with MSC.Nastran “ , 2nd MSC Worldwide Automotive User Conference, Dearborn, MI, Oct 2000.