Title:
Consistent Generation of FEA Meshes for Warped Thin-Walled Components Characterized by Geometric and Topological Complexities

Authors:
Theogenes de O. Maia, tmaia2@uwo.ca, Western University
O. Remus Tutunea-Fatan, rtutunea@eng.uwo.ca, Western University
Ryan Gergely, ryan.gergely@gm.com, General Motors
David A. Okonski, david.a.okonski@gm.com, General Motors

Keywords:
Warped Thin-Walled Part, Consistent FEA Mesh Generation, Difficult to Mesh Geometry

DOI: 10.14733/cadconfP.2023.271-275

Introduction:
The primary goal of this study was to develop a method to create a mesh of a given deformed part to be then used in structural analysis (FEA). For this purpose, the warped mesh will be derived from scans of real parts, and simulations of the assembly process will be used to determine the clamping forces required to close gaps between parts. This type of analysis is particularly relevant to ultrasonic welding operations.

When meshing solid parts characterized by a dominant 2D structure, the mid-surface meshing technique and shell elements are well-established and documented approaches [1], [2]. Furthermore, when comparing the structural behavior of components characterized by different warpage patterns, mesh consistency becomes a critical requirement. If the parts are meshed without enforcing the consistency, then the results obtained for a family of parts – essentially characterized by the same geometry but different warpage patterns and magnitudes - cannot be compared in a reliable manner. Furthermore, the topological complexity of some of the CAD models prevents their meshing when attempted with commercial meshers. In such cases, any downstream structural analysis on them is virtually impossible. Along these lines, for parts made of composite materials, nominal dimensions can vary depending on the manufacturing process. For example, compression molding - commonly used in the automotive industry for thermoplastic-matrix composites - causes the material to flow within the mold. As the part's geometry becomes more complex, the fiber concentration and orientation are affected, and this in turn leads to local changes in mechanical properties, surface defects, and volume defects such as shrinkage and warpage, all identified and reported in literature [4], [3]. As such, when relying on automated/commercial meshing tools – regardless if based on solid or mid-surface meshing approaches - it is practically impossible to guarantee that the same part, produced in at least two different warping patterns, will be consistently meshed, thus allowing side-to-side comparisons between nodes and elements of interest. By contrast, when mesh consistency is introduced, local information can be extracted and statistically analyzed thus enabling guided improvements and alterations of the process parameters.

The use of ultrasonic welding is a desirable joining process since it allows for facile integration of thin-walled composite components into more complex assemblies. Nonetheless, since these components are often warped/deformed, it is important to know and/or predict if the welding head can withstand the additional loading determined by the non-nominal shape of the parts to be subjected to ultrasonic welding [5]. To estimate or predict the force required to close the gaps between warped parts to be ultrasonically welded, factors such as the local relative warpage (between joining parts) and local material properties play a significant role. As the number of local factors to be considered when attempting to assess the magnitude of the assembly force - such as warpage and material properties - increases, it becomes increasingly important to maintain mesh consistency. For this purpose, a constant thickness was initially assigned as a section property in Abaqus using the surface scan of warped parts. Unfortunately, this approach could not account for variations in nominal thickness. Additionally, as shown in Figure 1, when working with a 3-part assembly, the contact between interacting surfaces may not always be detected. The mid-surface meshing technique was also attempted, but it was found that it is practically impossible to represent the warping pattern by solely relying on the mid-surface. As such, a non-automated/user-controlled solid mesh technique was selected as the optimal approach, because this allows for consistent representations of any warping pattern when starting from the same mesh.

Main Idea:

An overview of the method developed for the consistent warping of FEA meshes is shown in Fig. 1. The starting point of the framework are two virtual objects: a solid mesh (generated by automatically meshing the "ideal" nominal/unwarped CAD geometry) and a warped tessellated surface (obtained by scanning the actual molded part that will inevitably be warped). The scanned surface will include all geometric features of the solid part (regardless if scanned on its "A" or "B" side) but it will be completely lacking the thickness information. Thus, the thickness is assumed to be at the nominal values at this time. If the scanned surface is aligned/registered to its nominal counterpart, its deformation/warpage pattern can be outlined by means of qualitative color map-based comparisons.

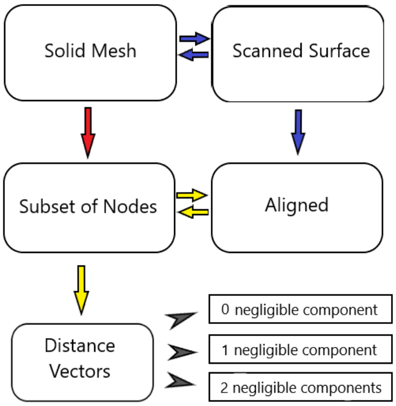


Fig. 1: Overview of the framework.

This qualitative information has to be translated into local and quantitative information. The latter could be described as a warping vector field since it associates each point with a vector that captures the local amount of deformation to be applied to the nominal mesh in order to match the deformed shape. To accomplish this, it is necessary to select nodes (locations) at which the vector field will be applied. This selection can be made at least in two ways: either by selecting a subset of nodes where most mesh discrepancies are present or by selecting a subset of nodes that are uniformly distributed within the geometry.

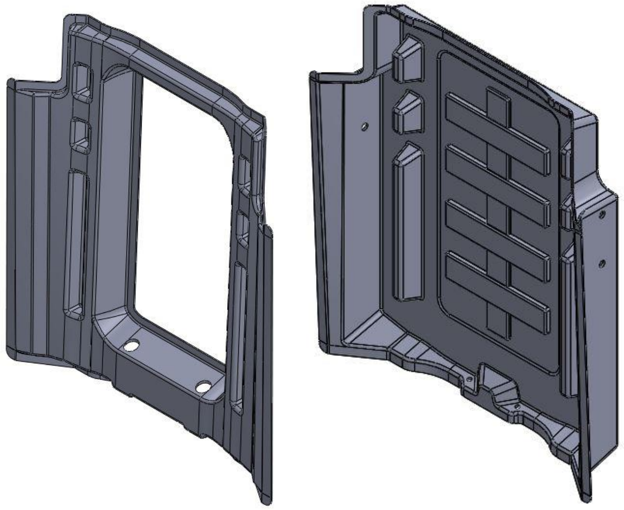
At a later point, a quantitative comparison will be made by calculating the distances/deformation vectors between the nodes and the warped surface, the selected nodes belonging to a subset of the solid mesh. The nodes analyzed on the unwarped mesh have to maintain their relative mapping/correspondence with respect to warped surface to which the distance is measured in order to avoid erroneous deviation vectors. Additionally, by properly aligning the part with the global coordinate system and considering the actual geometry of the part, the direction of the discrepancy can be estimated, leading to the differentiation of three main subsets of nodes as follows: i) nodes with extensive discrepancies along one axis and with negligible deviations along the other two axes, ii) nodes characterized by excessive discrepancies along two axes but with negligible values along the third, and iii) nodes with no negligible discrepancy values.

The aforementioned deviations/discrepancies can be measured automatically by means of commercial software such as PolyWorks. By applying metrics - such as the shortest distance - 3D deviation vectors can be determined. Then, according to the aforementioned subset differentiation criteria, some of the component(s) of the deviation vector could be ignored. As a result, a maximum of seven categories of nodes can be identified: three groups of vectors with two components ignored (X, Y, Z), three groups of vectors with one component ignored (XY, XZ, YZ) and one group in which none of the components (XYZ). However, certain groups can be eliminated due to the initial alignment and the lack of information about the thickness of the scan.

Then, after eliminating some components, the resulting distance vector can be used as displacement boundary conditions in an Abaqus simulation. Once the simulation is finished, the deformed part can be compared with the warped scanned surface using either a qualitative or quantitative method. If the comparison is not satisfactory, adjustments can be made at various phases to improve the outcome. Several of the commonly encountered workflow scenarios were:

- The mesh and scanned surface can be better aligned before measuring local distances in order to improve the accuracy;
- The alignment between the mesh and the global coordinate system is subject to adjustments;
- The selected nodes can be modified in terms of quantity, position, or both;
- For certain points, the components of the vector can be redefined and this might prompt a change of the subset of nodes.

Any of these workflow scenarios can be repeated successively and iteratively until the warped mesh reaches an acceptable tolerance level.



(a) (b)

Fig. 2: Sample geometries: (a) seatback inner (SBI) and (b) seatback outer (SBO).

Results:

The developed method was applied to the meshing of two sample geometries called seatback inner (SBI) and seatback outer (SBO) (Fig. 2). The acceptable deviation of the resulting warped mesh was set to 0.5 mm. Owed to the selection criteria that were detailed above, only a very limited subset of the node of solid mesh were selected for further mesh warping purposes. More specifically, out of the total of 281889 nodes that were present in the solid mesh of the SBO, only 1990 (0.705%) were selected to apply the deformation field and create a warped solid mesh. Similarly, only 884 (1.05%) of SBI's nodes were used to create its deformed mesh.

The starting mesh that was created in the nominal geometry can be consistently warped according to the deviations captured in the one-sided scan of the fabricated part. This enabled the use of the warped mesh in subsequent finite element simulations aiming to predict the clamping forces required to close the gaps between deformed SBO/SBI pairs. Figure 3a depicts the typical boundary conditions employed for this purpose: orange arrows indicate clamping points whereas point A is the first ultrasonic welding point. Of note, ultrasonic welding is always applied once the two parts have been fixture/clamped together. Nevertheless, the order in which clamps are being closed as well as the order in which ultrasonic welding is applied play a significant role on the stress levels of the upper (inner) part primarily due to the continuously variable nature of the gaps to be closed by the ultrasonic weld.

To complete the assembly simulations, a computer with the following specifications was used: IntelR CoreTM i7-6700 CPU at 4.00GHz, 32 GB RAM, 64-bit Windows OS, 10 Abaqus tokens (6 CPU cores, without concurrent jobs). Figure 3b pictures the stress experienced by the parts after the fixing clamps were applied. A plot that shows the correlation between the local displacement and local pressure as they change over time is provided for welding point A (located at the bottom left flange) in Fig. 4. Since the area of the welding head is approximately 78 mm^2, the maximum amount of force yields around 1 kN. Furthermore, Tab. 1 shows the amount of time it took to complete the simulation for each point, which was calculated by subtracting the starting time from the ending time as provided by Abaqus. This information can be used for future comparisons and optimizations and to convey the fact that simulation time is highly dependent on the magnitude of the gaps to be closed at each step. In a sense, the larger gaps could take a significant amount of time in order yield convergent simulations (Simulation of gap closing for clamping purposes) whereas smaller gaps can be closed virtually faster (Simulation of gap closing for welding purposes).

Conclusions:

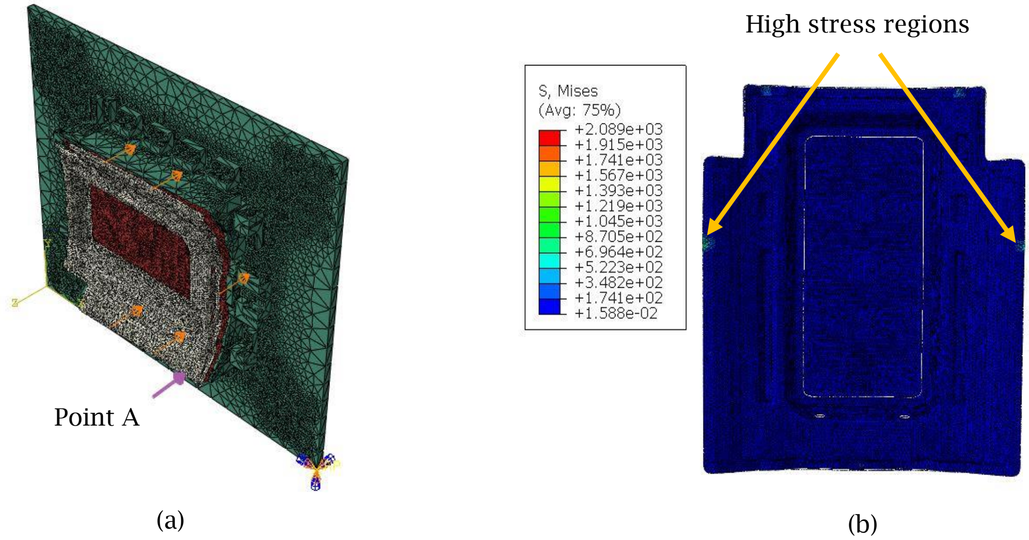


Fig. 3: Sample simulations of the assembly for warped components: (a) boundary conditions and welding point and (b) stress levels at clamping points (MPa).

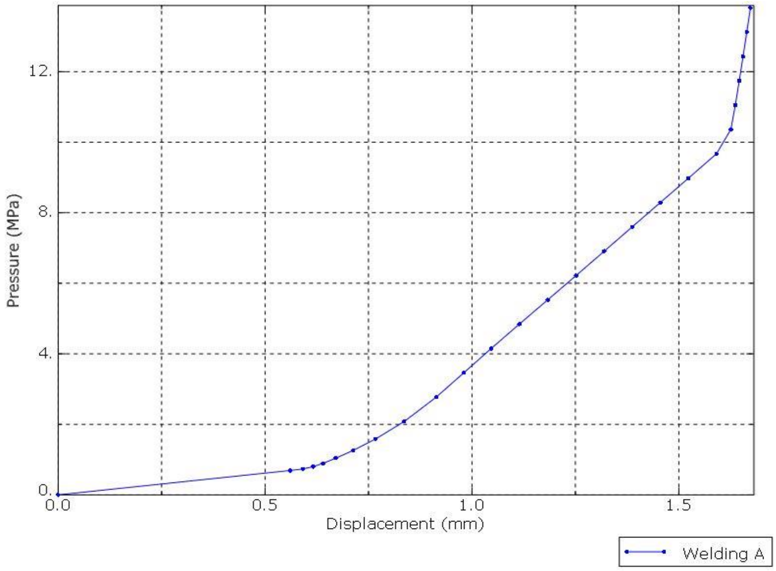


Fig. 4: Characteristic loading curve for point A.

In this study, the consistent deformation of meshes was achieved by means of a selecting a specific group of nodes. One of the most important advantages of the developed approach resides in that once the reduced subset of nodes was identified for a specific nominal geometry, it can be used to warp the nominal mesh according to the deformation field of virtually any physical/fabricated part. This is a consequence of the fact that the reduced subset of nodes is independent/invariant with respect to the warpage pattern of the parts as obtained from different part manufacturing/molding conditions.

Simulation Type	*Steps*	*Simulation Time (hh:mm:ss)*
Simulation of gap closing for clamping	1st+2nd+3rd clamp closing	03:43:03
	4th+5th clamp closing	04:29:11
	6th+7th clamp closing	03:15:18
Simulation of gap closing for welding	Bottom point on left flange	01:39:04
	Midpoint on right flange	01:14:53
	Midpoint on top flange	01:02:55

Tab 1: Sample simulation time for various assembly operations.

Nonetheless, one of the drawbacks of the developed method is represented by the amount of time needed to identify and select the subset of nodes to be included in the nominal mesh to be deformed according to the physical warpage pattern. At this time, this process was primarily manual and hence error-prone and user dependent albeit its automation could constitute a future development direction. Nonetheless, once the subset of nodes to which the deformation vector was applied was chosen, it can be warped to match any final shape of the molded part. Another limitation of this method is that it cannot consider variations in thickness of actual parts that are assumed to have a constant thickness throughout. While additional provisions can be made to address this concern as well, it can be assumed – at least for the time being and/or as a first/initial approximation - that thickness variations do not have a significant effect on the characteristic loading curves.

Overall, in addition of generating a qualitatively superior and relatively small-sized deformed mesh, the method is applicable to any thin-walled geometry. This method is also applicable to geometries that cannot be meshed by means of the algorithms that are integrated in the commercial FEA software. The typical examples in this category are topologically complex CAD models that are either a consequence of faulty modification operations and/or of incomplete/defective 3D scanning.

Acknowledgements:

The work presented in this study is the result of the collaboration between Western University (London, Canada) and General Motors (Warren, USA). Partial financial support was also provided by Natural Sciences and Engineering Research Council (NSERC) of Canada, Ontario Centers of Excellence (OCE), and General Motors of Canada.

References:

[1] Cifuentes, A.; Kalbag, A.: A performance study of tetrahedral and hexahedral elements in 3-D finite element structural analysis, Finite Elements in Analysis and Design, 12(1992, 313-318.

[2] Danielson, K. T.; Adley, M. D.; Williams, T. N.: Second-order finite elements for Hex-Dominant explicit methods in nonlinear solid dynamics, Finite Elem. Anal. Des., 119(C), 2016, 63–77. http://doi.org/10.1016/j.finel.2016.02.008

[3] Song, Y.; Gandhi, U.; Koziel, A.; Vallury, S.; Yang, A.: Effect of the initial fiber alignment on the mechanical properties for GMT composite materials, Journal of Thermoplastic Composite Materials, 31(1), 2018, 91-109. http://doi.org/10.1177/0892705716681400

[4] Song, Y.; Gandhi, U.; Pérez, C.; Osswald, T.; Vallury, S.; Yang, A.: Method to account for the fiber orientation of the initial charge on the fiber orientation of finished part in compression molding simulation, Composites Part A: Applied Science and Manufacturing, 100(2017, 244-254. https://doi.org/10.1016/j.compositesa.2017.05.021

[5] Villegas, I. F.: In situ monitoring of ultrasonic welding of thermoplastic composites through power and displacement data, Journal of Thermoplastic Composite Materials, 28(1), 2015, 66-85. http://doi.org/10.1177/0892705712475015