

# Accurate Simulation-Based Predictions of Assembly Force and Deformation for Warped Thin-Walled Components Characterized by Geometric and Topological Complexities

Theogenes de O. Maia<sup>1</sup>, O. Remus Tutunea-Fatan<sup>2</sup>, Ryan Gergely<sup>3</sup>, David A. Okonski<sup>4</sup> and Marco Nicolini<sup>5</sup>

<sup>1</sup>Western University, <u>tmaia2@uwo.ca</u> <sup>2</sup>Western University, <u>rtutunea@eng.uwo.ca</u> <sup>3</sup>General Motors, <u>ryan.gergely@gm.com</u> <sup>4</sup>General Motors, <u>david.a.okonski@gm.com</u> <sup>5</sup>Western University, <u>mnicolin@uwo.ca</u>

Corresponding author: O. Remus Tutunea-Fatan, rtutunea@eng.uwo.ca

**Abstract.** A large number of thin-walled parts used in industry are fabricated through thermal processes that often translate into deviations from their nominal geometry. These deviations – typically termed as warpage – might pose significant challenges to downstream assembly operations since the amount of force required to clamp the parts will be inevitably amplified by the magnitude of the gaps that are present between the parts to be joined. As such, the amount of assembly force has to be accurately predicted in order to size the assembly fixture correctly. Nonetheless, the accuracy of assembly simulations is often hindered by the fact that part models cannot be meshed due to their complex topology. To address this, the current study has developed a meshing method capable of circumventing the difficulties encountered by commercial FEA software. The use of this method has enabled the completion of the assembly simulations that have demonstrated that clamping and welding forces can be predicted with reasonable accuracy. The validation of the numerical results has been completed by means of a physical assembly fixture that was instrumented with load and displacement measuring sensors.

**Keywords:** Warped Thin-Walled Part, Consistent FEA Mesh Generation, Difficult to Mesh Geometry, Assembly Operations, Prediction of the Assembly Forces **DOI:** https://doi.org/10.14733/cadaps.2024.998-1017

### **1** 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. 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 [6],[7]. 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 3D scan data of warping patterns of the same part, will be consistently meshed. Thus side-by-side comparisons between nodes and elements of interest are not possible. 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 [9]. 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.

Since 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, in this study a constant thickness was initially assigned as a section property in Abaqus using the surface scan of warped parts. Unfortunately, this approach does not account for variations in nominal thickness. Additionally, when working with a multiple 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.

Once the meshing of the warped parts was completed, assembly forces and deformations – characteristic to both clamping and ultrasonic welding operations – were predicted through numerical simulations and then compared against their counterparts obtained through physical experiments. The upcoming sections will detail each phase of this work.

### 2 PREPROCESSING AND WARPING SIMULATIONS

#### 2.1 Overview

To illustrate the process developed for the consistent generation of meshes, an assembly consisting of two demonstrator parts called seatback outer (SBO) and seatback inner (SBI) will be used. These two thin-walled parts - fabricated through a compression molding process - are to be subsequently joined by means of an ultrasonic welding process. For reference purposes, the dimensions of the bounding box for each of the two composite parts used were 540 x 480 x 98 mm.

To develop simulations capable of predicting the amount of forces and deformations required for the assembly operation with a reasonable degree of accuracy, one of the first obstacles to be overcome is related to the fact that commercial FEA software is often unable to mesh the topologically complex geometry of the nominal/undeformed part. Furthermore, even when possible, meshing the nominal part has little value for the aforementioned simulations to be performed on geometries that replicate the physical parts that are characterized by large warpage. To address this, the present section will detail the novel approach that was developed and used to mesh the warped thin-walled geometry of manufactured SBO and SBI components.

The workflow used for this purpose entails the following steps: (i) scan one side of the physical part; (ii) mesh the nominal part; (iii) deform the nominal mesh to match the warpage pattern of the physical part; (iv) compare the warped solid mesh with the scanned geometry. Figure 1 summarizes this process whose steps will be detailed in the upcoming sections.

#### 2.2 Digitization of the Physical Geometry

The geometry of the fabricated demonstrator parts was digitized with a laser by scanning one of the faces. Figure 2a illustrates a sample of the triangular mesh obtained. The initial assembly simulation tests assumed a conventional shell model for the two analyzed parts (SBO and SBI) and a solid model for the clamping fixture involved. However, the simulation trials performed revealed that while



Figure 1: Phases of warped mesh generation.

the contact between solid and shell elements can be detected (fixture/SBO pair), the contact cannot be defined between two shell models (SBO/SBI pair) [1],[2].

This prompted the need for an alternate meshing method capable to enable an appropriate use of the contact conditions between the elements of the assembly. Furthermore, the alternate meshing method had to be applicable to topologically complex geometries to be obtained through single face scanning of the two thin-walled components involved in the assembly.



**Figure 2:** Nominal (grey/blue) vs real geometry (green/blue): (a) scanned geometry and (b) overlaid nominal and scanned geometries.

## 2.3 Mesh Deforming Procedure

#### 2.3.1 Node Grouping

The developed meshing method relies on the deformation of an undeformed/unwarped solid mesh generated for the nominal/undeformed part geometry that is warped according to a deformation field. This field is comprised of deformation vectors that quantify the difference between the position of a point on the nominal geometry and its post-manufacturing correspondent (on the warped geometry). A typical discrepancy between nominal and warped geometries is depicted in Figure 2b. In other words, the application of the deformation field to a set of nodes belonging to the nominal mesh will allow the mesh to be deformed in a manner that matches the scanned/warped geometry. Similar techniques were developed for computer graphics purposes [5],[8]. Nonetheless, while sharing certain similarities, the mesh deformation method to be detailed in this study has two important differences with respect to computer graphics techniques: i) relies on 3D, rather than 2D elements and ii) has the deformation targets set by the final shape of the warped part rather than by physical force-deformation constraints. Of note, the 3D elements used in this work are also defined internally in a different manner than the one that is typically used for graphical mesh deformation purposes.

While various techniques could be used to determine the deformation field, this work will present an approach in which the mapping between the undeformed and deformed position of the nodes is facilitated by grouping. The groups of nodes to be created such that the deformation vector corresponds to the one, and at most two of the principal axes of the global coordinate system. As Figure 3 suggests, the groups of nodes to be created are primarily dictated by the negligible/small deformation vector component in the directions of the remaining one or two principal axes as determined by the geometry comparison software. In more general terms, the number of groups of nodes to be formed will depend on the complexity of the geometry and that of the warpage field to be captured. For instance, while the three groups of nodes shown in Figure 4 were sufficient to define SBO's deformation field, SBI geometry required an additional group that included deformation vectors characterized by non-null X and Z components. The registration procedure used to align the nominal and warped geometries was identical to the one presented in more details in [4].

Several recommendations can be made with respect to the mesh warping process: 1) it is advisable to maintain a uniform density of the selected points, 2) selection of nodes located too close to fillets or corners should be avoided because they do not provide reliable deformation vector measurements (the software is incapable of producing coherent measurements in geometry

transition zones), 3) the mapping between deformed and undeformed locations of the nodes needs to be assessed carefully since the correspondence is not always present (Figure 4) and 4) no nodes should be selected from the relatively flat central regions since their contribution to the deformation field is minimal.



(c)

Figure 3: Node-grouping procedure for SBO geometry: (a) X group, (b) Y group, and (c) Z group of nodes.

After the distances between all points and the scanned surface were measured, the list that contained the coordinates and vectors of each point was used to generate displacement boundary conditions to be inputted into finite element models. Importantly, the remaining degrees of freedom for the deformation vectors were not dictated (e.g., for the X group, displacement was only prescribed in the X direction, whereas the other DOFs - Y, Z, and all 3 rotational axes - were free).

The following sections will detail the main distinctions between the warping models associated with the two analyzed geometries.

#### 2.3.2 Deformed Mesh Model for Seatback Outer

Once the nodes were grouped, the nominal mesh was deformed according to the displacements captured by the deformation field. The purpose of this simulation was to obtain a mesh that matches the warped shape of the actual part. The material was



**Figure 4:** Inexistent mapping between warped and unwarped geometries (Z-group nodes) caused by large deformations.



Figure 5: Deforming the SBO mesh: (a) boundary conditions, (b) resulting mesh, and (c) overlay between warped mesh and scanned part.

assumed to be uniform, isotropic and characterized by the following properties: density =  $1.45 \times 10^{-9}$  ton/mm<sup>3</sup>, Young's Modulus = 14300 MPa, Poisson's ratio = 0.35. While the material model used represents a simplification of the glass fiber reinforced composite that was used to fabricate the part, the results obtained suggested that the aforementioned material model simplification/substitution did not introduce any significant errors. Future research efforts could enhance the accuracy of the material model, primarily by adding non-uniform and anisotropic characteristics [6].

Once the matrix of boundary conditions was created and the simulation was completed (Figures 5a, 5b), the resulting mesh was exported out in a triangular mesh format whose top layer/surface was then compared against the scan of the warped part (Figure 5c). Quantitative descriptors of this comparison are shown in Table 1. In brief, the data shows a mean close to zero table shows a mean close to zero (-0.015 mm) with 93.4% of nodes between -0.385 mm and 0.355 mm with respect to the target/scanned surface ( $\pm 2$  standard deviations). If the screening range is extended further, then 100% of nodes were found at a distance between -0.57 mm and 0.54 mm ( $\pm 3$  standard deviations). This suggests that the SBO warping procedure has generated a mesh that matches the shape physical part with a good accuracy.

While the utilized node grouping procedure remains a manual process at this time, the superior match between the deformed mesh and fabricated part makes the developed procedure worthwhile. It is also important to note that this accuracy was obtained by applying the deformation field to a small subset of the solid mesh nodes. More specifically, out of the 281887 nodes, only 1990 (0.705%) were required to obtain the warped mesh model. The total simulation time was close to 30 seconds.

Total points evaluated	281887
Mean of deviation (mm)	-0.015
Standard deviation (σ)	0.185
<b>±σ</b> (199896 points, 70.914%)	+0.170 / -0.200
<b>±2</b> σ (263300 points, 93.406%)	+0.355 / -0.385
<b>±3</b> σ (281887 points, 100.000%)	+0.54 / -0.57

 Table 1: Quantitative comparison between warped mesh and scanned SBO.

### 2.3.3 Deformed Mesh Model for Seatback Inner

The same procedure used to deform the SBO mesh was also used to warp that of SBI. As noted above, the presence of inclined surfaces drove the need for a fourth group of nodes (with nonzero X and Z components) to be included in the simulation. By contrast, only the three categories of

nodes shown in Figure 4 were required for SBO. Figures 6a and 6b depict the boundary conditions and the result of the mesh deforming simulation.

Similar to the previous section, Figure 6c encompasses a qualitative comparison between the deformed mesh and scanned geometry. However, in this case the preset  $\pm 0.5$  mm deformation range outlines the fact that some regions of the geometry are "out of bounds" with respect to the preset deviation tolerance. While further refinements of the model are possible (to be achieved by increasing the number of selected nodes in the "out of bounds" areas), this was not within the scope of the current study. Nor was it necessary, since the focus of the downstream investigation/assembly operation was solely on the outer and inner flanges of the part. These flanges were within the range of the preset tolerance.



**Figure 6:** Deforming the SBI mesh: (a) boundary conditions, (b) resulting mesh, and (c) overlay between warped mesh and scanned part.

The quantitative descriptors of the overlay (Table 2) reveal that while the deviation mean remains low (-0.065 mm), SBI will have 97.9% of nodes between -0.577 mm and 0.447 mm with respect to the target surface (two standard deviations). This mesh warping accuracy was obtained by applying the required deformation/ displacement field to 1.05% of the total number of mesh nodes (840 of 83574). The total simulation time was approximately 70 seconds.

Total points evaluated	83574
Mean of deviation (mm)	-0.065
Standard deviation ( $\sigma$ )	0.256
<b>±σ</b> (51867 points, 62.061%)	+0.191 / -0.321
<b>±2</b> σ (81791 points, 97.867%)	+0.447 / -0.577
<b>±3</b> σ (83574 points, 100.000%)	+0.703 / -0.833

Table 2: Quantitative comparison between warped mesh and scanned SBI.

### 3 CLAMPING SIMULATIONS

Simulations were developed replicating the sequence in which the clamps are to be applied on pair of fixtured SBI and SBO. The goal of these simulations is to predict the amount of force to be applied either by the welding tool or by the clamps. If the two parts would be manufactured at (or close to) their nominal shape, then the amount of assembly or welding forces to close the gap will be zero (or at minimal values). However, due to the large warpage accumulated in the geometry of the fabricated parts [4], the magnitude of the clamping forces increases. To recreate a virtual replica of the actual assembly process – according to which the operator will place SBO in the fixture and then SBI on top of it before closing the first clamp – simulations were performed in two stages. At first, SBO geometry was positioned on the fixture in order establish the initial contact between them. After that, a second simulation was conducted to mimic the placement of the SBI over SBO. To accurately predict assembly forces, a special attention was dedicated to contact regions as well as their definition. More details on this topic will be provided in the upcoming sections.

## 3.1 Contact Definition

At first, a simplified CAD model of the metal fixture – obtained through the elimination of numerous non-essential components - was imported into the simulation software. The fixture was defined as a discrete rigid part and with a rather coarse mesh (Figure 7) since its deformation and stress levels were negligible compared to those of the composite parts to be assembled. Furthermore, the only active components of the fixture were the central blocks where the contact with SBO flanges was present.

## 3.1.1 SBO/Fixture Contact

In an attempt to reduce the computational cost associated with automatic software-detected contact, surfaces prone to come in contact during assembly were preselected and defined as master (fixture) and slave (SBO) (Figure 8). These roles were attributed according to the rigid or deformable characteristic of the corresponding simulation component. Additionally, the surface-to-surface discretization method was utilized. In this approach, the master and slave geometries are considered

in the region of contact, not just at the contacting nodes. Although some node penetration may occur, there will be no substantial regions of "master" nodes penetrating the slave surface, which represents an enhancement over the node-to-surface formulation. The parameters used for this contact interface were: i) tangential behaviour with a friction coefficient of 0.3, ii) normal behaviour with a "hard contact", allowing separation after initial contact and iii) damping coefficient of 0.1 when clearance is 0.0.

## 3.1.2 SBI/SBO Contact

A similar idea was employed for the SBI/SBO pair. Each surface was defined as a mesh-type, including all adjacent element's faces. Figures 8 and 9 illustrates the surfaces defined in SBO as master and in SBI as slave. Although this interface was defined between two parts with identical material characteristics, the same contact parameters outlined in the previous section were used. The contacts between SBO and SBI defined in this work were identical to the ones presented in a previous study [3].



Figure 7: Meshed clamping fixture.



(b)

Figure 8: Contact surfaces for the SBO/fixture pair: (a) master surfaces on fixture and (b) slave surfaces on SBO.



Figure 9: Contact surfaces for the SBI/SBO pair: (a) master surfaces on SBO and (b) slave surfaces on SBI.

### 3.2 Clamping Sequence

This section presents and analyzes the clamping pattern employed to secure the parts and the boundary conditions applied to the model, aiming to replicate the actual assembly conditions. The main outcomes of the assembly simulation are the final relative position of both parts with respect to the fixture and with respect to themselves and the variation of the gap along the flanges to be welded.

The clamping pattern employed in the simulation matches one of the possible options available in practice (Figure 10). While more clamping scenarios exist, the one used here is regarded as sufficient for the "proof of concept" of the investigation methodology presented in this study.

The upcoming subsections will detail the simulations associated with each clamping step (Figure 11): a) close clamps 1-3, b) close clamps 4 and 5 while keeping clamps 1-3 closed, and c) close clamps 6 and 7 while keeping clamps 1-5 closed.



Figure 10: Clamping point locations.



Figure 11: Clamping steps: (a) close 1-3, (b) close 4-5 (red dots), and (c) close 6-7 (red dots).

To simulate the clamping sequence, the restart option of Abaqus was utilized. The clamping steps were defined as dynamic and the boundary condition was determined based on the available resources for post-clamping comparisons. Although pressure or force measurements at the clamping heads could not be obtained, the vertical distance needed to bring both parts into contact and close the gap was measurable in both virtual and real assembly scenarios.

In the real assembly, gaps were measured with calipers whereas node distance queries were used for the same purpose in the simulation software. Consequently, a boundary condition based on node displacements was implemented. The displacement amount to be applied was determined by measuring – for each clamping area – the distance between a node in the back surface of SBI and a node in the vertical block of the fixture. While this measurement produced a 3D vector, only the Z-component was retained. Even though the head of the clamp was made from a compliant material (rubber), it was assumed – for simplicity – that clamping did not result in any material compression.

All flanges have a nominal thickness of 2 mm. Therefore, after determining the Z-component of the distance vector, SBO flange thickness was subtracted to establish the Z-direction boundary condition, while allowing all other DOFs to remain free. Table 3 displays the Z-measurements

obtained through this process as well as boundary conditions along Z applied to all clamping points. Since SBI was not uniformly warped along its flanges and its initial position was not perfectly parallel to the fixture (as a manual and thus error-prone operation), it is reasonable to expect that symmetrically positioned points will not have identical gap values.

Clamping Point	Z Measurement (mm)	Z BC Applied (mm)
1	18.95	U3=-16.95
2	21.02	U3=-19.02
3	16.84	U3=-14.84
4	20.79	U3=-18.79
5	16.61	U3=-14.61
6	20.20	U3=-18.20
7	17.74	U3=-15.74

#### Table 3: Gap measurements and displacements applied at each clamping location.

#### 3.2.2 Second Clamping Step

The use of the restart option guarantees the continuity of the results. As indicated in Figure 11b, the second step involves closing of the clamps 4 and 5 while maintaining 1-3 closed. Same as in the first step, clamp closing was equivalent with a Z movement constraint (while the other DOFs were left unrestricted).

### 3.2.3 Third Clamping Step

This step corresponds to the closing of the remaining clamps (6 and 7) while clamps 1-5 were kept closed. Same as for the previous step, the restart option was used (Figure 11c and Table 3). After the completion of the third clamping step, the assembly is fully clamped such that the post-clamping analysis can be performed.

#### 3.3 Post-Clamping Results

To evaluate the accuracy of the simulation, the gap between SBI flanges to fixture was measured – both virtually and physically – after all clamps were closed. The gap measurement was performed for all flanges that were externally accessible on the physical setup. For all measurements, the fixture blocks were used as references. The same distances were also measured in the virtual setup. Table 4 presents a summary of the comparisons between virtual and physical gaps as assessed for the points indicated in Figure 12.

Of note, for all evaluation points that coincide with clamp locations (B, C, F, J, M and, O) the SBI to fixture gap was assumed to coincide with the SBO thickness. As a result, no difference was calculated for these points. Since for all other points no relative difference between physical and virtual gaps larger than 5% was observed, it was



Figure 12: Gap assessment locations.

concluded that the virtual setup is accurate enough to proceed with the subsequent phases of analysis.

Points	Physical Assembly Gap (7 clamps closed) (mm)	Virtual Assembly Gap (7 clamps closed) (mm)	Relative Difference (mm)	Relative Difference (%)
Α	4.71	4.48	-0.23	-5%
В	SBO thickness	SBO thickness	-	-
С	SBO thickness	SBO thickness	-	-
D	4.84	4.88	0.04	1%
E	4.06	3.95	-0.11	-3%
F	SBO thickness	SBO thickness	-	-
G	2.70	2.72	0.02	1%
Н	2.59	2.53	-0.06	-2%
I	3.86	3.84	-0.02	-1%
J	SBO thickness	SBO thickness	-	-
K	2.34	2.40	0.06	2%
L	2.51	2.38	-0.13	-5%
М	SBO thickness	SBO thickness	-	-
Ν	5.18	5.05	-0.13	-3%
0	SBO thickness	SBO thickness	_	-

 Table 4: Virtual and physical gap comparisons.

## 4 WELDING SIMULATIONS

The welding simulations to be presented next were focused on the subsequent stage of assembly, during which SBI and SBO were to be joined/welded at predetermined locations.

## 4.1 Possible Welding Points

Figure 13 identifies all possible welding locations on SBI surface. As indicated, welding can be done either on the external flanges (A, B, C, D, S, T, AA, BB, Z, R, X, J) or on internal flanges (L, M, N, O, P, Q) or on internal features termed as "pockets" or recesses (E, F, G, H, U, V, W, Y). While all these possible welding points could be investigated, the current study will focus on just several locations to be thoroughly analyzed: A (bottom left flange), AA (middle right flange) and S (middle top flange).

## 4.2 Simulation Parameters and Set-up

The best starting point of the welding simulations is constituted by the previously completed clamping scenarios. To achieve this, a new step was created with the same attributes as the previous ones, including dynamics, the number of increments as well as the initial time increment. To facilitate the contact between SBO and SBI at welding points, a boundary



Figure 13: Possible welding locations.

condition capable to output a reaction force or pressure on the area of contact was added. The magnitude of this force or pressure represents a limiting factor for the welding head. This limiting factor can be used in the sense the welding device can withstand only a certain amount of pushback force. Nonetheless, unlike the case of clamping simulations for which concentrated forces were determined by node displacements, welding simulations required the extraction of force, stress or

pressure that can be inaccurate in case of concentrated outputs. Therefore, the BCs used for clamping could not be used for welding simulations [9] and a pressure load was utilized instead (the total force was applied over a preset area). The region on which the pressure was applied was determined by the size of the welding head. More specifically, the load was distributed over certain faces matching the location of the welding head and the maximum force was recorded for plotting purposes. In terms of the general simulation setup, large displacements were allowed. Also, the large number of contacts defined between the two parts and the fixture has significantly increased the nonlinearity of the simulations that in turn translated in longer run times (Table 5).

Once the virtual welding simulations completed for points A, AA and S, the amount of reaction force was collected as a simulation output (Table 6).

Welding Point	Simulation time (hh:mm:ss)
А	1:39:04
AA	1:14:53
S	1:02:55

Table 5: Simulation time for closing of the gaps at welding points.

Welding Point	Maximum Force Measured (N)
Α	1127.66
AA	1561.82
S	1150.43

**Table 6:** Maximum force recorded at welding point locations.

After the completion of the simulation, the characteristic curves - capturing the dependence between the closing force and the displacement/gap at the analyzed welding point location - were extracted and they will be discussed in the subsequent sections.

### 4.3 Results

The characteristic curves for the three analyzed welding points are presented in Figure 14. It is important to note that the characteristic curves were obtained by assuming that all other seven clamps were already closed. The differences between the amount of displacement recorded at each of the three welding locations are correlated with the size of the gap that was measured once the clamping was completed - point A in Table 4 corresponds to welding point A, point K corresponds to welding point AA and point N corresponds to welding point S. Along these lines, Table 7 presents an estimation of the vertical displacement required to close the gaps at welding locations. The estimated values were obtained by accounting for the fact that the nominal thickness of SBO is 2 mm.

Gap Measurement	Welding Point	Vertical Gap (mm)	Estimated vertical displacement (mm)
А	А	4.48	2.48
К	AA	2.40	0.40
Ν	S	5.05	3.05

Table 7: Estimated vertical displacement required.



Figure 14: Characteristic curves at welding locations: (a) A, (b) AA, and (c) S.

The curves in Figure 14 provide insight on the gap closing behavior at each welding location. In one respect, if the estimated force/pressure is larger than the force required to close the gap, then compressive loads will start to develop at the analyzed point. Furthermore, the visual analysis of the characteristic curve reveals that several distinct sections exist, with each of section associated with a specific pressure/displacement slope. For instance: while point A is associated with three different slopes that are connected through smooth/gradual transitions, point AA is characterized by only two distinct slopes connected through a sharp change. By contrast, the two slopes that are present at point S have relatively close values.

The behavior observed at point A is predictable since the stiffness of the warped material opposing its downward bending increases with each contact until bending converts into material compression once both gaps (SBO to fixture and SBI to SBO) are closed. The behavior present at point AA implies that the first inflexion point – associated with the SBO-SBI contact – is suppressed, likely because the geometry of SBO used in this simulation was no significantly warped and as such both contacts (SBI-SBO and SBO-fixture) occurred almost simultaneously. Finally, in case of point S, the pressure applied during the simulation was insufficient to close the gap such as the third section of the characteristic curve (material compression) was not present.

#### 4.4 Validation

### 4.4.1 Experimental Setup

As previously presented, the assembly of thin-walled parts to be joined consists of SBI and SBO. To facilitate welding, the two assembly components – whose geometries are severely affected by thermal-induced warpage as a result of their upstream manufacturing process (Figure 15) - have

to be affixed together with clamps to be applied prior to welding. In order to validate the results of the virtual setup described in Section 3 and the first part of Section 4, a physical setup was designed and fabricated.

In brief, the experimental setup used for validation consists of: i) assembly fixture with a modular format capable to allow the facile modification of the assembly/welding points (Figure 16) and ii) load and displacement sensors capable to ensure a concurrent measurement of the two assembly parameters that constitute the primary input for the characteristic curves presented in Section 4.3.



Figure 15: Assembly gaps.



Figure 16: Fixture for force and gap measurements: (a) CAD model and (b) physical embodiment.

The solution used for load/displacement measurements includes an S-type load cell (RB-Phi-204 500Kg from Phidgets) and a linear variable displacement transducer (LVDT) type PR750-1000 from Macro Sensors). Both sensors were coupled with signal conditioners: DMA2 (from Interface) for the load gage and UCM (from Penny + Giles) for the LVDT. As Figure 17 suggests, the load cell will measure any forces created by the upward movement of the vertical rod contacting the assembly. A vertical spring attachment was used to counteract the weight of the rod assembly. The displacement was measured on a direction parallel to that of the load. The two measurements were caused by the same upward motion of the main rod but they were decoupled in order to increase their accuracy. For this purpose, LVDT was mounted to a separate crossbar than the one involved in load measurements.

To acquire the data needed for the characteristic curves, the driving bolt was tightened with a maximum feed of 1 rot/sec. Following this, the sampling rate was chosen 10 times higher such that a measurement was taken at every 5  $\mu$ m of travel of the driving bolt (maximum closing speed). These values were selected in such a way to achieve a certain balance between the speed and accuracy of the force/displacement measurements. These values also ensure an acceptable computational load with respect to the amount of data to be processed.



**Figure 17:** Measuring module: (a) physical assembly, (b) CAD model, (c) load cell retention spring and (d) LVDT mounting.

The tip of the measurement rod is characterized by a circular area of 77.93 mm<sup>2</sup>. This value allows straightforward conversions of the gap closing load into pressure and this in turn facilitates direct comparisons between virtually and experimentally obtained characteristic curves.

#### 4.4.2 Experimental Results

Figure 18 presents a typical characteristic clamping curve obtained at point F (Figure 13) for a specific SBO-SBI pair obtained through manufacturing trials. The *X*-axis indicates the distance from the fixture blocks with a zero reference (0 mm) set on their highest surface. The red vertical line delimitates the compounded thickness of the two parts. This means that all gaps are closed once the displacement reaches this level such that further increases of the clamping force will translate solely into compressive loads on the two parts. While the shape of the characteristic curve is strongly dependent on the assessment point, point F was selected for clarity reasons in a sense that the trends presented on the line can be explained with a reasonable degree of accuracy.

Since the tip of the force measurement module is initially not in contact with SBI, the force remains null for displacement/gaps larger than 6.5 mm. Once the tip of the rod contacted the upper surface of SBI, the load gauge started to record increasing values of the clamping force/pressure. In this case, the slope of the characteristic curve is dependent only on the stiffness of the SBI. Once SBI contacted SBO, the second inflexion point appeared on the curve and the change in slope can be explained by means of the increased stiffness of the assembly to be clamped (now consisting of both SBI and SBO). Finally, the third inflexion point corresponds to the moment when all gaps are closed and the three assembly components (fixture, SBO, SBI) are in paired contacts with each other (SBI-SBO and SBO-fixture). After all gaps are closed, the pressure at the clamping point builds up quickly as the material enters a predominantly compressive regime. It is also important to note that all five loading trials exhibited a similar behavior, and this suggests that both parts remained in their elastic deformation domain.

The behavior described above matches qualitatively but not quantitatively the one observed at welding point A (Figure 14a). The next section will attempt to provide insight on the possible causes of the observed discrepancies between the virtual and experimental characteristic curves.



Figure 18: Characteristic loading curve for point F; red line denotes the moment when all gaps are closed.

#### 4.5 Discussion

This section will be focused on a direct comparison between virtual and experimental results. The first sample in this category is point A (Figure 19a). As it can be observed, both curves include two inflexion points and three segments. This suggests that the underlying physical behavior captured by the two curves is similar in a sense that SBI moves down under the action of the welding head until SBI contacts SBO. After that, the pair of flanges moves together until all gaps are closed.

Nonetheless, while the pattern of variation of the two curves is relatively similar, the numerical values of the slopes involved are different and same applies to the magnitude of the clamping force. Several factors can be cited as possible causes for these discrepancies. Among them, it can be mentioned here that subsequent investigations revealed that virtual results were confounded by the non-uniform thickness of the flanges that was introduced by the initial geometry healing process. Furthermore, Figure 19a implies that while at point A all gaps were closed in the physical assembly after approximately 0.8 mm displacement, this state was achieved in the virtual setup only after 1.7 mm displacement was applied. To correct this discrepancy of approximately 0.9 mm, it is conceivable that an increased number of nodes selected at point A combined with a more appropriate warping procedure could be used. Finally, the difference in stiffness between the digital model and experiment can also be a consequence of the contact properties used for each interface as well as by the hypothesis of isotropic material properties. To tackle the final aspect, one potential enhancement could involve the examination of the inherent characteristics of the molded components followed by the subsequent mapping of these features onto the distorted model. This approach would account for anisotropy, a characteristic material property of the composite components.

A similar level of discrepancy is also noticeable between the characteristic curves associated with point AA (Figure 19b). Nonetheless, since the relative position of AA between clamps is different than that of A, a much stiffer behavior can be identified on the characteristic curve associated with the virtual model. Furthermore, the experiment revealed that when closing the gap at point AA, the parts can shift in a lateral direction, a phenomenon that was not present in the simulation. These



**Figure 19:** Comparison between experimental and simulation results at: point A, (b) point AA and (c) point S.

additional degrees of freedom that are present in the physical, but not the virtual experiment might be a consequence of the idealized assumptions made in the simulation with respect to friction and contact conditions. These additional degrees of freedom/movements that are present in the physical tests have inevitably led to the reduced stiffness that is identifiable on the associated characteristic curve. As a final comment on point AA, the physical displacement/gap was found to be about 2 mm larger than the virtual one and this could suggest that after all clamps were closed the physical parts warped more than their virtual counterparts. This increased gap could also be a consequence of the overall increased stiffness of the virtual assembly that has also translated into increased force values.

All these factors have likely also affected the closing behavior at point S whose virtual characteristic curve did not exhibit a second inflexion point and that could imply that the gap between the three components of the assembly remained open at the end of the simulation. This means that the maximum force applied was insufficient to close the gaps present in the virtual assembly. Of note, since the gap measured in the beginning of the virtual closing simulation at point S was within 5% of the real value measured on the physical setup, it can be speculated that the discordant virtual characteristic curve is a consequence of the combined effect of all closed clamps on the overall stiffness of the virtual assembly.

## 5 CONCLUSIONS AND FUTURE WORK

The main goal of this study was to develop a mesh deforming method that can be successfully applied to topologically complex and warped geometries associated with thin-walled components that are typically fabricated through thermal-based processes. The utility of the developed method was then demonstrated by means of a case study involving an assembly of two composite components that are to be joined by means of ultrasonic welding while being clamped together in a fixture.

In addition of generating a qualitatively superior and relatively small-sized deformed mesh, the method is applicable to any thin-walled geometry. As a limitation, the proposed approach relies on the manual determination of an initial subset of nodes and this is a rather tedious and time consuming procedure. However, it is important to note that this operation does not need to be repeated for any other warped instances of the same geometry for which the deformed mesh can be generated rapidly. Furthermore, the developed meshing method is unable – in its current form - to account for parts with variable thickness that could be a concern for certain categories of thinwalled components. As shown above, the method proved to be capable to generate warped solid meshes characterized by a 5% deviation with respect to the geometry of the manufactured part. While this represents an acceptable level of accuracy, further improvements can be brought by adding more nodes to the deforming subset. These supplementary nodes are to be added in certain regions of interest, typically those characterized by larger values of error.

Even if the proposed mesh deforming method yielded reasonable levels of error with respect to the geometry of physical/scanned part, the structural finite element analyses relying on this mesh proved to be more difficult to control. In other words, direct comparisons between virtual and physical clamping scenarios revealed that while the post-clamping gaps tend to remain within 5% of each other, the characteristic loading curves could be significantly different for some of the clamping/welding points (AA, S). As well, a rather large discrepancy between virtual and physical scenarios was noticed for values of the clamping force. In more general terms, the simulation overestimated the magnitude of the clamping force and this can be certainly attributed to a larger stiffness of the virtual assembly with respect to its physical counterpart.

Among the factors that can be cited for these discrepancies, the friction and contact properties between the two thin-walled components can be listed as having a major impact on their warping behavior. Since no data on composite-to-composite contact/friction is available in the surveyed literature, this study assumed values that are specific to plastic to plastic contact. Further errors were introduced by the inability of the virtual setup to replicate accurately the real contact between the clamping/welding tip and the loading patch defined the SBI geometry. To further elaborate on this, the loading patch was assumed loaded with a uniform amount of pressure applied constantly in the vertical direction, whereas in the real setup loading could be applied only on a fraction of that patch (due to the warped geometry) and – more importantly – lateral part shift is permitted even under the action of the clamping/welding load. The lack of the lateral part shift under the action of the clamping forces represents one of the most significant limitations of the virtual scenario since the inherent cancellation of a number of degrees of freedom leads to stiffness increases and therefore overestimations of the clamping/welding forces needed.

Evidently, any enhancements capable to address this issue would enhance the precision of the virtual assembly simulations that could be used to predict the effect of various clamping scenarios on the welding force, one of the important metrics associated with this type of assembly operation. Other possible improvements could attempt to automate the process of warping node selection, to enable the use of components with variable thickness, to enhance the material models used as well as to determine the actual properties of composite-to-composite contact. Additional improvements could also target the concurrent instrumentation of the clamps for force/displacement measurements such that the source of simulation errors can be better narrowed down.

In summary, while both the developed mesh deforming approach and the virtual assembly scenario remain perfectible, they represent one of the first attempts aiming to tackle the challenges

associated with the virtual simulation of the assembly performed on thin-walled (composite) components.

## ACKNOWLEDGEMENTS

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

*Theogenes Maia*, <u>https://orcid.org/0009-0008-0693-0109</u> *O. Remus Tutunea-Fatan*, <u>https://orcid.org/0000-0002-1016-5103</u> *Ryan Gergely*, <u>https://orcid.org/0000-0003-0351-9177</u> *David A. Okonski*, <u>https://orcid.org/0000-0003-0485-6749</u> *Marco Nicolini*, <u>https://orcid.org/0009-0001-5139-843X</u>

#### REFERENCES

- 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. <u>http://doi.org/10.1016/0168-874X(92)90040-J</u>
- [2] Danielson, K. T.; Adley, M. D.; Williams, T. N.: Second-order finite elements for Hex-Dominant explicit methods in nonlinear solid dynamics, Finite Elements in Analysis and Design, 119, 2016, 63-77. <u>http://doi.org/10.1016/j.finel.2016.02.008</u>
- [3] Martin, E.; Tutunea-Fatan, O. R.; Gergely, R.; Okonski, D.: Repeatability and accuracy of laser scanning-based reverse engineering for warped composite components, Computer-Aided Design and Applications, 18, 2021, 1018-1034. <u>http://doi.org/10.14733/cadaps.2021.1018-1034</u>
- [4] Martin, E.; Tutunea-Fatan, R.; Gergely, R.; Okonski, D.: Quantitative characterization of warpage for composite components, in Computer-Aided Design and Applications, 2021, <u>http://doi.org/10.14733/cadaps.2022.1093-1108</u>
- [5] Sahillioğlu, Y.: Recent advances in shape correspondence, The Visual Computer, 36, 2020, 1705-1721. <u>http://doi.org/10.1007/s00371-019-01760-0</u>
- [6] 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, 2018, 91-109. <u>http://doi.org/10.1177/0892705716681400</u>
- [7] 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. <u>http://doi.org/10.1016/j.compositesa.2017.05.021</u>
- [8] Van Kaick, O.; Zhang, H.; Hamarneh, G.; Cohen-Or, D.: A survey on shape correspondence, In: Proceedings of Computer graphics forum, Wiley Online Library, 2011, 1681-1707. <u>http://doi.org/10.1111/j.1467-8659.2011.01884.x</u>
- [9] Villegas, I. F.: In situ monitoring of ultrasonic welding of thermoplastic composites through power and displacement data, Journal of Thermoplastic Composite Materials, 28, 2015, 66-85. <u>http://doi.org/10.1177/0892705712475015</u>