ANA LYSIS OF MI LI T -MAT E R IAL BO ND E D A S S E M BL E S O N A NO - CO N FO RM IN G M E S H

Goldy Kumar
Email: gkumar3@wisc.edu
Spatial Automation Laboratory
Department of Mechanical Engineering
University of Wisconsin-Madison
Madison, Wisconsin, 53706

Vadim Shapiro
Email: vshapiro@engr.wisc.edu

ABSTRACT

Bonded multi-material assemblies arise frequently in design, manufacturing, architecture, and materials design. It is a common wisdom that finite element analysis of such assemblies usually requires all components to be represented by compatible finite element meshes; application of meshfree methods in such situations is often considered problematic due to the need to impose additional interface conditions. Neither approach scales to deal with realistically complex models arising in many applications.

We propose a simple extension of meshfree analysis on a non-conforming mesh for linear structural analysis of such multi-material assemblies. The method is simple, can be implemented within most FEA packages and does not require either compatible meshing or complex interface boundary conditions. Our numerical experiments demonstrate that computed results are in good agreement with known analytical and computational results for well studied multi-material bonded assemblies (lap and butt joints). We also demonstrate application of the proposed method to realistically complex assembly of a mounted sculpture that cannot be easily analysed by other methods.

1 Introduction

Assemblies of two or more solids, usually made from different materials and bonded together, are common throughout engineering application ranging from basic lap and butt joints to complex multi-material structures arising in bio-medical, consumer, automotive, aerospace, and many other industries [1–4]. In all these cases, the solid bodies are rigidly bonded together by some of the many possible physical processes, or by an additional adhesive material which geometrically forms a solid body in its own right. Structural analysis of such assemblies requires determination of displacements and stresses throughout the solids, as well as at the material interfaces which are characterized by jump conditions: discontinuities in strains, high stresses, and/or stress concentrations at the edges of the interfaces known as “edge effects” in laminates [5, 6].

Analytic, approximate analysis and experimental results are known for commonly used simple assemblies such as lap and butt joints [4, 7, 8] shown in Figures 4a and 14a respectively. Analysis of more complex multi-material structures requires numerical simulation, typically using finite elements analysis (FEA). An example of such a multi-material structure is shown in Figure 18a. It consists of three solids: a marble sculpture mounted on steel rods and supported by a resin epoxy base. Linear static analysis of the assembly under vertical gravitational forces is usually required to predict stresses and displacements, for safety of in-
stallation and conservation purposes. In order to use conventional FEA tools, such an assembly of three solids need to be converted into a mesh of finite elements (with constant material properties) that must conform to the material interfaces within the assembly. This mesh generation task often becomes a bottleneck, particularly when solids in the assembly come from different sources or applications.

The challenges associated with mesh generation led to numerous proposals for generalizing and extending FEA techniques to work on non-conforming meshes. These include use of enrichment functions [9–11], penalty functions or Lagrange multipliers [12–14], special solution structures [15], and many other techniques such as those based on Nitsche and fictitious domain approaches [16–18]. While many of these proposals aim for numerical stability and convergence of the solution, their implementation introduces a number of practical challenges and the adoption by commercial systems has been slow. See [19] for a recent survey and discussion of the issues related to modeling of jump conditions.

This paper explores a simple and pragmatic alternative to modeling of jump conditions at the material interfaces. In a nutshell, we solve the analysis problem on a non-conforming mesh. They include use of enrichment functions [9–11], penalty functions or Lagrange multipliers [12–14], special solution structures [15], and many other techniques such as those based on Nitsche and fictitious domain approaches [16–18]. While many of these proposals aim for numerical stability and convergence of the solution, their implementation introduces a number of practical challenges and the adoption by commercial systems has been slow. See [19] for a recent survey and discussion of the issues related to modeling of jump conditions.

The rest of the paper is organized as follows. Section 2 briefly discusses the material interface conditions in a typical rigidly bonded assembly, as well as their implementations in FEA using conforming and non-conforming meshes. In section 3 we compare the solutions on non-conforming mesh against analytic and conforming mesh FEA solutions for lap and butt joints. We demonstrate application of the proposed method to the sculpture assembly in 4 and finally conclude with a brief summary in Section 5.

2 Formulation and approaches

2.1 Bounded Assemblies and Material Interfaces

A typical bonded assembly is a finite connected union of rigid solids – components of the assembly, with distinct interior material properties, and whose boundaries intersect on material interfaces. A bonded assembly of two components A and B is schematically illustrated in Figure 1. In this case, the material interface ∂Γ is the intersection of the components’ boundaries ∂ΩA and ∂ΩB respectively. From a structural point of view, the two components may be considered in a state of static contact. Equivalently, a rigidly bonded assembly may be considered a single solid with a discrete heterogeneous material structure.

Outside the material interfaces, the boundaries of components in the assembly may be restrained and loaded as usual in a linear static analysis model, and we are interested in determining displacements, strains, and stresses throughout the assembly, including the points on the material interfaces. Since the bodies are rigidly attached, the displacement field \( u(x,y,z) \) must vary continuously throughout the assembly, and the displacements within the components A and B must match on the interface boundary \( ∂Γ \):

\[
\frac{∂u}{∂Γ_A} = \frac{∂u}{∂Γ_B} \tag{1}
\]

Furthermore, since the entire assembly must be in equilibrium, surface tractions \( T \) across the material interface must vanish at every point.

\[
T_A + T_B = 0 \tag{2}
\]

This effectively means all stresses must be continuous across the interface.

The interface boundary conditions (1) and (2) are naturally satisfied in both strong and weak formulations of the linear elasticity problem [23, 24]. Consequentially, all finite element methods satisfy these conditions automatically without any additional effort – to the same extent as they satisfy these conditions in analysis of a single isotropic solid. The difference in material properties in the adjacent components in the assembly has several consequences. Increased stresses and strain discontinuities properties occur across the material interface, and stress concentrations often manifest themselves near the edges and corners of
the interface boundaries. It should be intuitively clear, that without explicit enforcement of these “jump conditions” at the material interfaces, the finite element solutions may approximate them only in the limit of mesh refinement. Below we consider this process for FEA on conforming and non-conforming meshes respectively.

2.2 Analysis on a conforming mesh

Finite element analysis on a mesh that conforms to the material interface is the most straightforward and preferred method for handling multi-material assemblies – when such meshes can be constructed. In this case, the solid meshes of the individual mating components are compatible (i.e. the nodes of the meshes coincide) on the material interfaces (see Figure 2a). Constant material properties can be assigned to the elements depending on which side of the material interface they lie, and the finite element analysis proceeds as usual. Recall that in a typical FEA scenario, equilibrium (of forces and moments) is enforced directly at the nodes, but not across inter-element boundaries or even within the elements [23] where equilibrium holds only on an average or integral sense over the element volume. Thus, strictly speaking, the traction conditions 2 are enforced only approximately. Stress field continuity across the element boundaries could be maintained with additional effort, for example by employing mixed finite elements [23] where both displacement and stress field are the primary variables.

The main problem with this approach is the difficulty of generating the required meshes that must conform not only to every solid in the assembly, but also to every material interface. The latter requires that the 2D mesh of every material interface serves as boundary for two incident solid meshes. The additional constraint that the two mating solid meshes agree on a common set of nodes presents challenges even for simple assemblies. Figure 2b shows one such mesh that was automatically generated in SolidWorks [25] in an effort to adapt to the very simple material interface. This task quickly becomes unmanageable for complex assemblies with larger number of components and detailed geometry.

An alternative approach to meshing an assembly is to mesh each solid component independently, and then impose a set of additional contact constraints on the meshes at the material interface. The additional constraints include various contact, slip, and stick conditions [26,27]. Methods for enforcing such constraints include mortar elements [28,29], Lagrange multipliers or penalty methods [27,30]. The main problem with this approach is that it transforms a relatively simple deterministic linear static problem into a much more complex non-linear contact problem that may require heuristic and iterative solution.

2.3 Analysis on a non-conforming mesh

In an effort to bypass the above difficulties of compatible meshing, we could mesh the bonded assembly as a single solid, essentially ignoring the material interfaces. No special treatment is needed for those finite elements that are fully contained within individual components. When a finite element intersects the material interface, we will refer to it as an interface element and slightly modify the solution procedure as follow. Instead of resolving and enforcing the jump conditions within the interface elements, we simply recognize that different points may fall within distinct components and use the corresponding material properties. Material properties come into picture during numerical integration for computing stiffness coefficients and equivalent body load, and for stress computations in the post-processing phase after displacements are computed.

Stiffness coefficients are computed by summing the contributions of individual element $K_e$:

$$ K_e = \int_\Omega B^T [\eta_i] D B [\eta_j] d\Omega, $$

where $\eta_i$ are the basis functions; following the widely used notation in FEA literature [23,24], we use $B$ to denote the matrix of derivatives, also known as the strain-displacement matrix, so
that \( \varepsilon = B[u] \), and \( D \) for the stress-strain matrix so that \( \sigma = D\varepsilon \).

The contribution of body forces to consistent element load is given as:

\[
R_e = \int_{\Omega_e} [\eta_j] F d\Omega_e, \tag{4}
\]

where \( F \) is the body force which can be a function of material property. The integration is carried out numerically using Gauss quadrature which amounts to sampling and taking a weighted sum of the basis functions and material properties at a finite number of Gauss points.

Figure 3b illustrates the modified integration procedure for a typical interface element. Every quadrature point \( p_i \) is tested for membership against the components in the assembly in order to determine the corresponding material property \( D(p_i) \) and \( F(p_i) \). This integration procedure effectively results in weighted averaging of the material properties in interface elements. The weight of each material is roughly proportional to the volume percentage occupied by that material in the interface element.

It should be clear that the continuity of displacements and stresses is maintained at all points of the material interface, but the jump conditions are not satisfied exactly. Instead, they are essentially spread over the interface elements and the supports of the corresponding basis functions. As the interface elements get smaller, this averaging error decreases and approaches the jump conditions, but only in the limit. In Section 3, we carry out a detailed numerical study of known solutions for lap and butt joint assembly using conforming and non-conforming mesh.

2.4 Meshfree FEA on non-conforming meshes

In principle, the proposed approach may be implemented within any existing finite element solution. For our numerical experiments we chose a meshfree Solution Structure Method (SSM) method [19], rooted in the classical work of Kantorovich [31] and Rvachev [32–34] and implemented as Scan&Solve method in [20].

In a nutshell, Scan&Solve implements the finite element method on a non-conforming mesh of multivariate B-splines \( \chi_i \) that covers the solution domain \( \Omega \). The essential (displacement) boundary conditions are enforced by modifying the basis function into a set of admissible basis functions \( \eta_i \) by setting

\[
\eta_i = \omega_u \chi_i, \tag{5}
\]

where \( \omega_u \) is a non-negative sufficiently smooth function that vanishes on the boundary \( \Gamma_u \) where essential boundary conditions are indicated. In principle, a variety of basis functions, including classical FEA shape functions, radial basis functions, trigonometric polynomials can be used, but B-splines on a uniform orthogonal grid are particularly popular due to their simplicity and attractive computational properties [35–37]. Once the basis functions are constructed, the numerical solution procedure follows the usual finite element formulation and computations. See [19, 20] for additional details and discussion.

The method is easily extended to perform linear static analysis on multi-material bonded assemblies with material averaging as proposed above. For integration purposes, the role of interface element in Figure 3 is played by the support of the individual basis function \( \eta_i \). The combination of the SSM meshfree method and material averaging is particularly attractive because it eliminates the needs for any conforming meshing and allows analysis of geometrically complex assemblies, such as the sculpture mount in Figure 18a.

3 Verification of Results

Material interfaces in a rigidly bonded assembly are zones of special interest as high stresses can initiate cracks and failures. It
is clear that computations on a non-conforming mesh lead to approximation errors in displacements and stresses at the points of the interface. In this section, we compare accuracy of the results computed by the meshfree Scan&Solve software against solutions from analytical models and solutions computed by SolidWorks [25] on a conforming mesh. Specifically, we chose lap and butt joints for our study because they are the most common and well studied multi-material bonded assemblies. At the same time, they exhibit most of the challenges in modeling of rigidly bonded material interfaces, including stress concentrations on the edges which must be reasonably approximated by any simulation method.

### 3.1 Simple Lap Joints

A simple lap joint consists of two substrates joined together by an adhesive as shown in Figure 4a. Application of axial loads induces shear stress $\tau$ in the adhesive region. Figure 4b shows the profile of the shear stress in the overlap region based on the model developed by Volkersen [4]. The maximum adhesive shear stress when the adherents are made of same material is given by

$$\tau = T \sqrt{\frac{G_a}{2E_s t \eta}}$$  \hspace{1cm} (6)

where $G_a$ is the adhesive shear modulus, $T$ is the shear loading, $E_s$ is the elasticity modulus of the adherent, $t$ is the thickness of the adherent and $\eta$ is the thickness of the adhesive. The maximum shear stress occurs on the edges of the overlap. Its value depends on the length of the overlap in the joint, up to some maximum overlap length.

This model assumes that the adherents are only in tension but the adhesive shear stresses induce moments in the adherents as well. This often results in “peeling stresses” on the edges.

A more general model for stress analysis of lap joints in a mixed loading condition was formulated by Bigwood and Crocombe [7] and is illustrated in Figure 5. With some simplifying assumptions maximum shear stress $\tau_T$ in the adhesive layer is estimated to be

$$\tau_T = \frac{1}{2} T \sqrt{\frac{G_a(1 - \nu_s^2)}{2E_s t \eta}}$$  \hspace{1cm} (7)

The estimated value is half the value calculated by Volkersen because of the introduction of moment and is considered to be a better estimate of the maximum shear stress.

Next we compare stress values obtained from FEA analysis using conforming mesh with the analytical model above. The FEA package used for this purpose was SolidWorks [25]. The reference model in Figure 4a ignores moment due to the eccentricity of the axial forces. SolidWorks permits analysis of a structure with comparatively small unbalanced load by applying so called Inertial Relief, wherein it automatically applies forces to counteract these unbalanced loads. The small moment generated due to eccentric loading in the lap joint model was neutralized using this feature. FEA model and the geometric and material properties of the lap joint being analysed is shown in Figure 6. The deformed lap joint due to loading is shown in Figure 7. The shape of the deformed lap joint matches qualitatively with the shape in Figure 8 from [38] and the shear stress profile in the adhesive region (Figure 9b) also agrees with the profile in Figure 4b. The shear stress in the adhesive region (Figure 9b) is maximum on the edges and is almost constant in the middle.
of the overlap region. The maximum shear stress values for the lap joint in Figure 6 predicted by Volkersen (6), Bigwood (7) and Solidworks are 117MPa, 55.1MPa and 130MPa respectively. Apparently, the maximum shear stress value computed by FEA is higher than the values calculated from both analytical models, presumably due to simplifying assumptions.

Finally, we compare results computed using a non-conforming mesh (Scan&Solve) with the results from conforming mesh FEA (SolidWorks). Deflection obtained from Scan&Solve for a lap joint with dimensions and boundary conditions used in Figure 6 is shown in Figure 10, indicating a deformation pattern qualitatively similar to that obtained from SolidWorks (Figure 7).

For a detailed numerical comparison, we use a different lap joint with dimensions in Figure 11. Reducing the size of the adherents relative to the size of the adhesive region allows to increase mesh resolution at the material interface for the same number of basis functions. Load is applied in the direction of the line joining the midpoints of faces A and B (Figure 11a) to eliminate any unbalanced moment. Computed displacement and the von Mises stress values on an edge of the top and bottom adherent (marked in Figure 11b) are plotted in Figures 12 and 13 respectively. Plots show values for different mesh resolution from Scan&Solve compared against result from SolidWorks which was computed with roughly 34,000 linear finite elements. The degrees of freedom in Scan&Solve and SolidWorks are not directly comparable due to difference in basis functions and
FIGURE 12: Plot Comparing displacement on the edges (Figure 11) for different resolution from non-conforming mesh with results from SolidWorks

meshing. For example, in SolidWorks, there were a large number of elements (17000) near the edges using adaptive meshing, giving more accurate values for the displacements. By contrast, Scan&Solve uses uniform size for all basis functions. Still the trend is very clear: as the number of elements is increased, the computed displacements in Scan&Solve converge to the values obtained from SolidWorks. Maximum error in displacement (at 47.5mm in 12b) is 4.8% (0.126 mm in Solidworks and 0.12 mm in Scan&Solve) which is less than 5% and can be considered in acceptable range.

Von Mises stress values from Scan&Solve are also close to those computed in SolidWorks, except in the adhesive region. Since adhesive layer is comparatively very thin, large number of elements are required for more accurate results. However the stress concentrations are correctly identified and as mesh resolution is increased, results converge everywhere including the adhesive region.

3.2 Butt Joints

In a typical butt joint, two adherents are joined together using a layer of adhesive and is acted upon by forces in the axial direction as shown in Figure 14a. It can also be subjected to shearing and torsional loads. Axial loading, in addition to normal stress, also leads to shear stress in the adhesive region if adhesive has lower Young’s modulus when compared to adherents. The simple analytical model for a cylindrical butt joint assumes that the radial and circumferential strains in the adherent and the adhesive are zero, in which case the radial stress, \( \sigma_r \), and circumferential stress, \( \sigma_\theta \), are given by:

\[
\sigma_r = \sigma_\theta = \frac{\nu}{1 - \nu} \sigma_z,
\]

where \( \nu \) is Poisson’s ratio of the adhesive and \( \sigma_z \) is the applied axial stress [39].

There are other analytical models but none of them are sufficiently accurate due to various simplifying assumptions. In particular, they do not take into account the presence of stress concentrations as suggested by [40]. The magnitude of these stress concentrations was observed to increase with increasing dissimilarity in the material properties of the components of the joint [8]. Because of these limitations, FEA models have been used to study the stress-strain behavior of butt joints. Plots from study done by Sawa [8] are shown in Figure 15a.

The butt joint configuration used to validate results from conforming mesh FEA is shown in Figure 14b and the geometric and material conditions are shown in Figure 15 with \( h_2 = 10\text{mm} \), \( E_1 = 2.00e + 11\text{N/m}^2 \), \( v_1 = 0.3 \) and a uniformly distributed axial force of \( F = 1\text{MPa} \). The two adherents have different material properties from each other and from the adhesive as well. This leads to different stress distribution at the lower and upper interface.

The plot from [8] for axial stress values from the center line to the corner on the upper and lower interface is shown in Figure 15a. Stresses are constant within the adhesive component but there is a stress concentration on the edge of the interface.

Copyright © 2012 by ASME
Stress concentration value increases with increase in $E_1/E_2$ ratio. Figure 15b shows the plot of axial stress values computed by Scan& Solve on a non-conforming mesh for the same butt joint (Figure 14b). The values are close but, more importantly, the nature of the stress distribution matches with the results from [8]. Higher $E_1/E_2$ ratio leads to higher stress concentration as expected. The computed average stress is higher than predicted, most likely due to insufficient degrees of freedom near the interface. Results from SolidWorks for the same conditions give similar results and trends (Figure 16).

Next we study the convergence of displacement value at the interface but for a modified assembly. Instead of having a regular butt joint, we used a two component assembly (Figure 17). This provided higher degrees of freedom at the interface for the
same number of basis functions without changing the nature of the problem. The assembly is made of two bonded 50x20x1 mm	extsuperscript{3} block made of epoxy and aluminium respectively. The bottom plate is grounded and a uniform pressure of 100 MPa is applied on the top face. Colormap for von Mises stress (Figure 17a) shows high stress near the interface. Figure 17b shows plot of the resultant displacement of the points on the interface for different resolutions compared against results from SolidWorks. Displacement values from both conforming and non-conforming FEA analysis are in good agreement and the curves are almost overlapping.

4 Analysis of sculpture assembly under gravity

We now show that the proposed approach for analysis of bonded assemblies scales to complex real world applications, such as analysis of the mounted sculpture shown in Figure 18a. The complete assembly consists of five different components bonded together (18b). The marble statue is mounted on a reinforced concrete pedestal using two steel pins. An epoxy base is used to rest the statue on the pedestal. The boundary of the statue is a union of 12017 surfaces while the epoxy base is made of 146 surfaces. The steel pins are modeled by regular cylinder and pedestal is a simple rectangular block. The boundaries of the components match at the material interfaces but are subjected to variable model precision and numerical accuracies. The complexity of the interface between epoxy base and the statue is shown in the top right of Figure 18b. The detailed boundary conditions are shown in Figure 19. Generating a conforming mesh for such an assembly is a challenging, tedious, and error-prone process that usually requires manual intervention and heuristic steps.

With the proposed approach, a conforming mesh is not required. The whole assembly is immersed into one common uniform rectangular grid that determines the number of basis functions (linear B-splines) and the resolution of the computed solution. In Scan& Solve, the restraints on the bottom surfaces of the epoxy base and the steel pins are imposed by modifying the basis functions using distance fields to the restrained boundaries, in the spirit of the Kantorovich method [20]. When the support of a basis function intersects more than one component, it becomes an interface element and its quadrature points are classified using point membership test against the components in the assembly. During the integration process, the material property at each quadrature point is assigned based on the outcome of the point membership test. Summing contributions of all quadrature points within an interface element amounts to averaging of the material properties.

To demonstrate the power of the proposed approach, we an-
analyzed the assembly at relatively low resolution of 10,000 basis functions. For maximum utilization of the available degrees of freedom, in this this experiment, we assumed that the pedestal can be treated as a ground. The computed von Mises equivalent stresses for the assembly are shown in Figure 20. As expected, relatively high stresses are observed at the interfaces and the interface edges. For example, stresses above 95 psi are clearly visible near the interface of epoxy and marble statue (Figure 20a). The highest stress values are observed near the interfaces of the pins and the epoxy base with the magnitude of 383 psi (Figure 21b).

In order to see the qualitative effect of presence of multiple materials in the assembly, we analysed a hypothetical non-physical assembly where all components are made of the same material (marble). As expected, the location of the max value of von Mises stress moved to the edge of the bottom of the restrained epoxy base, while its value dropped to about 196 psi. See Figure 21. This is consistent with the fact that high stresses at the material interfaces arise only due to material differences and are not sensitive to geometric noise and errors.

The above experiments demonstrate feasibility and applicability of the proposed approach for the analysis of complex rigidly bonded assemblies, but are not meant to predict the actual values of stresses and displacements. As with all methods, verification of the correctness of the computed results requires careful study and additional analyses at multiple resolutions and with varying assumptions. Such a study is outside the scope of this paper.

5 Conclusion

We have shown that in many applications rigidly bonded multi-material assemblies can be effectively analysed via simple material averaging on a non-conforming mesh. The implementation is straightforward, can be implemented within any FEA software, and scales to handle realistically complex multi-material assemblies. The averaging of the material properties in interface elements leads to loss of accuracy in the solution, but the errors appear to be acceptable even at relatively low resolutions and experimental results indicate rapid convergence when mesh is refined.

Our preliminary results lead to a number of open issues and further opportunities. It is clear that the accuracy of computations may be improved with adaptive meshing and multi-resolution computations. It is also conceivable that some applications will demand explicit jump conditions to be incorporated at the material interface. Identification of such situations and a careful further study of practical trade-off between the two approaches warrants a further investigation.
(a) Front of the sculpture: magnified material interfaces between pins, epoxy base, and marble statue.

(b) Back of the sculpture: overall distribution of stresses.

FIGURE 20: Von Mises stress distribution in the sculpture assembly. Stresses above 95 psi are visible in red.

(a) Single material

(b) Multi-material

FIGURE 21: Maximum stresses in the assembly of the sculpture when analysed as a) hypothetical all-marble assembly b) multi-material assembly of marble statue, epoxy base and steel pins

Acknowledgements

This research was supported in part by the National Science Foundation grants CMMI-1042211, CMMI-0856778 and CMMI-1029553. The model of assembly representing installation of the sculpture was provided by Jerry Podany and BJ Farrar of Getty Museum. The authors gratefully acknowledge many helpful discussions with Michael Freytag of Intact Solutions, who also provided programming interface and support for Scan&Solve software. The responsibility for errors and omissions lies solely with the authors.

REFERENCES


