An approach for treating contact surfaces in Lagrangian cell-centered hydrodynamics

Nathaniel R. Morgan *, Mark A. Kenamond, Donald E. Burton, Theodore C. Carney, Daniel J. Ingraham

X-Computational Physics Division, Los Alamos National Laboratory, P.O. Box 1663, Los Alamos, NM, USA

ABSTRACT

A new method is presented for modeling contact surfaces in Lagrangian cell-centered hydrodynamics (CCH). The contact method solves a multi-directional Riemann-like problem at each penetrating or touching node along the contact surface. The velocity of a penetrating or touching node and the corresponding forces are explicitly calculated using the Riemann-like nodal solver. The contact method works with material strength and allows surfaces to impact, slide, and separate. Results are presented for several test problems involving both gases and materials with strength. The new contact surface approach extends the modeling capabilities of CCH.

1. Introduction

We wish to simulate the motion of interacting deformable bodies that may be in full, partial, or no contact. In the general case, at the microscopic level in the boundary layer between the bodies, frictional and heat transfer mechanisms could be involved. In this work we simplify to the frictionless case in which the only coupling mechanism is the normal momentum component for the portion of the bodies in contact. Each of these bodies will be modeled with a Lagrangian mesh. Numerical schemes that address this interaction are termed contact surface algorithms.

Contact methods have been developed and used in Lagrangian staggered-grid hydrodynamic (SGH) calculations for many years. Early examples of contact methods are discussed in Wilkins [37] and Cherry et al. [7]. Hallquist et al. [17] provides an overview of multiple contact algorithms used in various Lagrangian SGH codes dating back to HEMP [37]. Of particular interest, Hallquist et al. [17] describes the contact surface scheme used in TOODY [31] and later implemented in DYNA2D [36]. The contact method of TOODY uses a master–slave approach. The goal of this approach is to treat the nodes on the contact surface in a manner similar to an internal node. The physical properties of the slave surface are interpolated to a ghost mesh (termed phony elements in [17]) that overlays the slave zones. The physical properties are interpolated from the slave surface to the ghost zones using surface area weights. The surface area weights are equal to the ratio of the ghost zone surface area to

* This is an open-access article distributed under the terms of the Creative Commons Attribution-NonCommercial-No Derivative Works License, which permits non-commercial use, distribution, and reproduction in any medium, provided the original author and source are credited.

* Corresponding author. Tel.: +1 5056670031.
E-mail address: nmorgan@lanl.gov (N.R. Morgan).
the surface area of the master surface. The contact surface method for nodal-based Lagrangian cell-centered hydrodynamics (CCH) presented in this paper will use surface area weights similar in concept to those in TOODY. Following the area fraction approach of TOODY may seem retrospective; however, using surface area weights naturally extends to the new CCH methods that solve a Riemann-like problem at the node of a zone [10,24,25,3].

Lagrangian CCH differs significantly from the SGH approach. The differences arise because the CCH method solves the conservation equations for the zone on a single control volume. Lagrangian CCH was first proposed by Godunov et al. [15,16] and later by Ruppel and Harlow [1] resulting in the CAVEAT code [1]. The CCH approach in CAVEAT solved a Riemann problem at the center of the zone face. The nodal velocity was calculated from the neighboring face velocities. Recent work in Lagrangian CCH has focused on Riemann-like solutions at the node. Despres and Mazeran [10] were the first to introduce a method that solved a Riemann-like problem at the node. Maire et al. [24,25] improved upon the nodal solution approach developed by Despres and Mazeran [10]. Burton et al. [3] extended the seminal works of [10,24,25] by proposing a new nodal Riemann-like method that handles stress tensors and ensures the viscous stress tensor is symmetric.

In this work, a multi-directional Riemann-like problem is solved at every node in the mesh including along the contact surface. The Riemann-like problem for the nodes away from a contact surface use the nodal solution approach of Burton et al. [3]. A similar Riemann-like problem is solved at the penetrating or touching nodes along the contact surface that takes into account the appropriate contact physics. The Riemann-like problem at a penetrating or touching node includes information from both surfaces. The contact surface approach maps the properties from the contacting nodes to the penetrating or touching node. The corner impedance, corner velocity, and corner stress of the nodes on the contacting surface are mapped to the penetrate or touching node using surface area weights. The surface area weights are equal to the ratio of the contacting surface area of the penetrating or touching node. The mapped properties are used in the nodal Riemann-like problem. The process is repeated for every penetrating or touching node along the contact surface. We do not adopt a master–slave approach. Every penetrating or touching node on each side of the contact surface uses the same Riemann-like solver.

The layout of the paper is as follows. Section 1.1 defines the nomenclature and notational conventions used in the paper. Section 2 discusses the finite volume conservation equations, presents the nodal solution approach, and provides an overview of the CCH solution methodology. The new contact surface approach is discussed in Section 3. Test problem results are in presented in Section 4, and the results of a bi-metallic shaped charge calculation and an exploding cylinder into a plate calculation are presented in Section 5.

1.1. Nomenclature

The nomenclature used in this paper follows the work in [3,4], and it is illustrated in Fig. 1. The stress, velocity, total energy, density and internal energy are respectively \( \sigma, u, j, \rho, e \). Vectors and tensors are shown in bold type. The physical quantities at the zone center are denoted with a subscript \( z \). The physical quantities projected to the node are defined as corner quantities and they are denoted with a subscript \( c \). The details of a zone corner are provided below. The nodal Riemann velocity is denoted with a superscript * and a subscript \( n \). The corresponding Riemann stress is defined in the zone corner so it is denoted with a superscript * and a subscript \( c \). The time is discretized using the second-order Runge–Kutta method and the time levels are denoted with a superscript \( n, n + \frac{1}{2} \), or \( n + 1 \) respectively.

![Fig. 1.](image)
A control volume around the zone is used with the evolution equations and a control volume at the node is used with the multi-directional Riemann-like problem. The control volumes are decomposed into triangles, where each triangle is termed an iota. The surface area of an iota face is \( A \) and it extends from the node to the center of the zone face. Each iota face is denoted with a subscript \( i \). The outward surface normal unit vector of the iota face is \( \mathbf{n}_i \). The outward surface area vector of the iota face is \( \mathbf{N}_i \), where \( \mathbf{N}_i = A \mathbf{n}_i. \) The iota nomenclature follows other Lagrangian hydrodynamic publications, specifically \([3,4]\); likewise, a similar decomposition is used in other Lagrangian CCH works \([10,24,25]\). Our motivation for decomposing the zone into smaller subsets is two fold. First, the decomposition allows the algorithm to be extended to arbitrary polygonal grids. Secondly, the multi-directional Riemann-like problem is solved at the node using the iota faces connected to the node. The control volume used for the multi-directional Riemann-like problem consists of all the iota faces connected to a node, which is denoted by \( i \in p \). Likewise, the evolution equations are updated using the Riemann values on all the iota faces in the zone, which is denoted by \( i \in z \).

The multi-direction Riemann-like problem at the node requires inputs such as impedance, velocity and stress. The Riemann-like problem inputs and results are defined in the zone corner, except the Riemann velocity. The Riemann velocity is defined at the node and it is constant over all zone corners around the node. The nomenclature for the zone corner is as follows. A zone corner quantity results are defined in the zone corner, except the Riemann velocity. The Riemann velocity is defined at the node and it is constant over all the iota faces connected to the node, and the Riemann stress is constant over the zone corner, and to emphasize the region where the quantities are defined. All Riemann-like problem inputs and connected to the node. The motivation for using a zone corner is to emphasize the Riemann-like problem inputs are constant over the zone corner, and to emphasize the region where the quantities are defined. All Riemann-like problem inputs and results are defined in the zone corner, except the Riemann velocity. The Riemann velocity is defined at the node and it is constant over all zone corners around the node. The nomenclature for the zone corner is as follows. A zone corner quantity operating on an iota surface area is denoted by a subscript \( c(i) \). For example, the projected velocity that affects iota surface area \( i \) is \( \mathbf{u}_{p(i)} \). Similarly, the Riemann stress, which is in the corner \( c \), operating on iota face \( i \) is \( \mathbf{\sigma}_{c(i)} \).

### 2. Cell-centered hydrodynamics

The discrete Lagrangian equations for conservation of mass, momentum, and total energy are

\[
\frac{\Delta M_j}{\Delta t} = 0, \quad \text{(1)}
\]

\[
M_j \frac{\Delta \mathbf{u}_j}{\Delta t} = \sum_{i \in z} (\mathbf{N}_i \cdot \mathbf{\sigma}_{c(i)})^{n+\frac{1}{2}}, \quad \text{(2)}
\]

\[
M_j \frac{\Delta \mathbf{a}_j}{\Delta t} = \sum_{i \in z} (\mathbf{N}_i \cdot \mathbf{\sigma}_{c(i)} \cdot \mathbf{u}_{p(i)})^{n+\frac{1}{2}}, \quad \text{(3)}
\]

where \( M \) is the mass and \( \Delta \) is the change between \( n \) and \( n + 1 \). The summations are over an arbitrary number of iota faces around the zone. Each iota face will have a corresponding Riemann velocity and Riemann stress. The Riemann velocity and corresponding stress operating on an iota face in the summation is denoted with a subscript \( p(i) \) or \( c(i) \) respectively. The Riemann velocity is constant over all the iota faces connected to the node, and the Riemann stress is constant over the zone corner. The nodal position, \( \mathbf{x}_p \), is updated using

\[
\frac{\Delta \mathbf{x}_p}{\Delta t} = (\mathbf{u}_p)^{n+\frac{1}{2}}. \quad \text{(4)}
\]

The Lagrangian volume rate of change of the zone is

\[
\frac{\Delta V_z}{\Delta t} = \sum_{i \in z} (\mathbf{N}_i \cdot \mathbf{u}_{p(i)})^{n+\frac{1}{2}}. \quad \text{(5)}
\]

A challenge with all Lagrangian hydrodynamic schemes (CCH and SGH) is accurately calculating the nodal velocity.

#### 2.1. Nodal Riemann-like problem

The cell–centered hydrodynamic approach in this paper solves a Riemann-like problem at the node \([3]\). Fig. 2 illustrates the nodal Riemann-like problem for a regular grid. The Riemann-like problem uses a control volume that extends from the node to the center of each connected iota face. A Riemann jump equation is written for each iota face.

\[
\mu_{c(i)} (\mathbf{u}_p - \mathbf{u}_{c(i)}) \mathbf{a}_{c(i)} \cdot \mathbf{n}_i = \mathbf{n}_i \cdot (\mathbf{\sigma}_{c(i)} - \mathbf{\sigma}_{c(i)}). \quad \text{(6)}
\]

The impedance of the zone corner is \( \mu \). Section 2.3 provides additional details on how the impedance is calculated. The unit vector \( \mathbf{a}_i \) points in the direction of the velocity difference between the corner and the node. The \( \mathbf{a}_{c(i)} \cdot \mathbf{n}_i \) term ensures the dissipation is in the direction of the jump in velocity. The unit vector \( \mathbf{a}_i \) is defined in each zone corner, hence, a subscript \( c \); likewise, the unit vector is constant over the two iota faces connected to the node. The unit vector \( \mathbf{a}_i \) is evaluated at the previous time step so that Eq. (6) is a linear equation. The Riemann-like problem at the node requires an additional equation to close the system of equations. Momentum conservation is enforced at the node to close the system of equations.
The nodal Riemann velocity is found by solving the system of equations involving the Riemann jump equations at each face and the momentum conservation at the node. The nodal Riemann velocity is

\[
u_p = \sum_{i,p} (\mu_{c(i)} | \mathbf{a}_{c(i)} \cdot \mathbf{N}_i | \mathbf{u}_{c(i)} - \mathbf{N}_i \cdot \mathbf{\sigma}_{c(i)}) / \sum_{i,p} \mu_{c(i)} | \mathbf{a}_{c(i)} \cdot \mathbf{N}_i |.
\]

The Riemann force, \(N_i \cdot \mathbf{\sigma}_{c(i)}\), acting on the surface is found via the Riemann jump equation (Eq. (6)), and \(N_i = A_i n_i\). The Riemann force is used in the discrete conservation of momentum (Eq. (2)) and total energy (Eq. (3)) equations. The nodal solver is valid for an arbitrary number of faces so the solver can be used on arbitrary polyhedra in both two and three dimensions.

2.2. Spatial projection

The inputs to the nodal Riemann-like problem are velocity and stress in the zone corner, \(\mathbf{u}_c\) and \(\mathbf{\sigma}_c\), respectively. As discussed in Section 1.1, the corner quantities are equal to the values projected to the node. The corner quantities are calculated via the zone-center quantities. The first-order accurate approach projects the zone-center quantity to the node, which neglects the gradient. The second-order accurate approach projects the zone-center quantity via a linear Taylor-series expansion.

\[
\mathbf{u}_c = \mathbf{u}_c + x_{pc} \cdot \nabla \mathbf{u}_c,
\]

\[
\mathbf{\sigma}_c = \mathbf{\sigma}_c + x_{pc} \cdot \nabla \mathbf{\sigma}_c,
\]

where \(x_{pc} = x_p - x_c\), which is a position vector between the zone center and the node. The gradient used in the linear Taylor-series expansion is limited to ensure no new extrema. The bar over the gradient denotes a limited gradient. The velocity gradient and limiting approach used in this paper were developed by Maire [25]. The stress gradient and limiter used in this work are described in Burton et al. [3]. The second-order accurate approach is used everywhere except at free surfaces and the contact surface. The solution at a contact surface or a free surface is first-order accurate in this paper, but could be extended to second-order. Future work will explore second-order accurate solutions at a free surface and a contact surface.

2.3. Impedance

The impedance of a zone corner, \(\mu_c\), is calculated using an approach based on the work of Dukowicz [12]. The details of the approach are as follows. The impedance in compression is equal to the density multiplied by the shock velocity, \(\mu = \rho_0 U\). The subscript, 0 on the density denotes the un-shocked state. The impedance in expansion is equal to the density multiplied by the acoustic wave speed, \(\mu = \rho_0 a\), which is the acoustic impedance. The following discussion will focus on the case where the zone is in compression. The shock velocity can be expressed as a linear function of the particle velocity \(u\). The linear relationship for the shock velocity is
where $C_0$ and $s$ are calculated experimentally. The particle velocity after the shock is $u^*$ and the particle velocity prior to the shock is $u_0$. The Dukowicz approximate Riemann solver uses a linear function for the shock velocity where the intercept is the acoustic wave speed, $C_0 \approx a$. The particle velocity of the un-shocked state is assumed to be equal to the corner velocity so $u_0 = u_c$. The impedance is a function of the velocity after the shock, $u^*$, which is the Riemann velocity; therefore, the Riemann equation on each iota face (Eq. 6) would be non-linear. To eliminate the non-linearity, the Riemann velocity in the impedance is assumed to be equal to the previous nodal Riemann velocity. Next, the impedance is a one-dimensional notion so the magnitude of the velocity difference is used in the impedance, $|u_p^* - u_c|/C_0$, $|u_p - u_c|$. Using these assumptions, the impedance of a corner becomes

$$U = C_0 + s|u^* - u_0|.$$  

(11)

If the zone is in expansion, then $s = 0$ and the corner impedance is equal to the acoustic impedance. If the zone is in compression, then a linear and quadratic dissipation term is produced when the impedance above is multiplied by the velocity difference, $(u_p^* - u_c)$, in the Riemann equation (Eq. (6)). In one dimension, the linear and quadratic terms are similar to those used in viscosity models for Lagrangian SGH $[33,22,28]$. The amount of dissipation added to the CCH calculation is a function of the velocity difference and the impedance. A larger impedance will produce more dissipation. The impedance in (12) is used for every calculation in this work and $s$ is the slope of $U-u$ relationship (Eq. (11)). The slope of the $U-u$ relationship can be found for a host of materials including metals in $[9,40,26]$.

### 3. Surface contact method

A contact surface approach can be viewed as a boundary condition that couples Lagrangian meshes together by enforcing contact-impact physics. When two separate meshes are in contact, they follow certain interface conditions, whereas, when the two meshes are not in contact they operate independently. One such interface condition is the continuity of the velocity and the stress in normal direction of the surface. The velocity and stress are discontinuous in the tangential direction of the surface. Another interface condition is the contact surface cannot support tension. In this work, these interface conditions will be enforced by solving a multi-direction Riemann-like problem in the normal and tangential directions of the contact surface.

![Diagram of contact surface](image)

Fig. 3. In the example above, a set of zones is traveling downward (arrow on left side) and collides with stationary zones 1 and 2. A contact surface separates the upper zones from the lower zones. One surface is connected to the penetrating or touching node and the other surface contains the contacting node set. For brevity purposes, a node that is touching or penetrating a surface will be called a penetrating node. In the example above, the contacting nodes are on the top surface and the penetrating node is on the bottom surface. The contacting node set are traveling downward and colliding with the penetrating zones. The velocity and associated forces at the penetrating node are calculated via a multi-directional Riemann-like problem at node.
3.1. Overview of method

A Riemann-like problem is solved at every node in the mesh. If a node is penetrating or touching another surface as shown in Fig. 3, then a Riemann-like problem is solved at the node that is based on the contact physics discussed above. A node that is touching or penetrating a surface will be called a penetrating node in this paper and is denoted with a subscript \( P_{pen} \). A Riemann-like problem is solved at the penetrating node in the normal direction of the contact surface and a different Riemann-like problem is solved at the penetrating node in the tangential direction of the contact surface. The nodes on the opposite surface of the penetrating node are termed contacting nodes. A contacting node is denoted with a subscript \( P_{con} \). The contacting node set is the set of nodes on the surface whose surface area overlaps the surface area of the penetrating node. The properties of the contacting nodes are mapped over to the penetrating node, and they are only used in the normal direction of the contact surface. The tangential direction calculation assumes frictionless slide so the properties of the contacting points are not included in that calculation.

The contact surface approach is derived by using a control volume at the node that extends from each point involved in the contact surface calculation to the center of the connecting face (i.e., an iota). The control volume is illustrated in Fig. 4. The nodal control volume for the normal direction includes the iota faces on the penetrating node and the iota faces of the contacting node set. The iota surfaces for the contacting node set may extend beyond the iota surfaces connected to the penetrating node. Area fractions are used to rectify the area discrepancy between the contacting iota surfaces and penetrating node iota surfaces. In contrast to the normal direction, the nodal control volume for the tangential direction only includes the iota faces attached to the penetrating node.

The Riemann velocity of the penetrating node in the normal and tangential directions is found via solving a set of Riemann equations on each iota face of the respective control volume and enforcing conservation of momentum at the node. Given a nodal Riemann velocity, the force on each iota face connected to the penetrating node can be found using the Riemann equations on each iota face of the respective control volume and enforcing conservation of momentum at the node.

3.2. Contact calculation in the normal direction

We wish to calculate the nodal Riemann velocity of the penetrating node and the associated forces acting on the node. This is achieved by solving a set of Riemann equations in the contact surface normal direction. Specifically, a Riemann equation, based on the work in [3], is written for all iota’s involved with the contact problem. The Riemann equation for the iota surfaces attached to the penetrating node is different than the iota surfaces on the contacting nodes. The sole difference is the Riemann equation on contacting nodes includes a surface area fraction to account for unaligned meshes. The Riemann equation for each iota face attached to the penetrating node is

\[
\mu_{(i)} (\mathbf{u}_{(i)} - \mathbf{u}_{(j)}) \cdot \mathbf{n}_{CS} = \mathbf{n}_{CS} \cdot (\sigma_{(i)} - \sigma_{(j)}) \cdot \mathbf{n}_{CS} - \mathbf{F}_{(i)} \cdot \mathbf{n}_{CS}.
\]

The contact surface normal vector at the penetrating node, \( \mathbf{n}_{CS} \), is used to select only the normal components. Section 3.6 discusses how \( \mathbf{n}_{CS} \) is calculated. The impedance in the zone corner is \( \mu_{(i)} \). The unit vector \( \mathbf{a}_{(i)} \) points in the direction of the velocity difference between the corner and the node. The term \( \mathbf{a}_{(i)} \cdot \mathbf{n}_{CS} \) ensures the dissipation is in the direction of the jump in velocity. The directional unit vector, \( \mathbf{a}_{c} \), is evaluated at the previous time step to make Eq. (13) linear. The velocity and stress are continuous in the normal direction of the contact and discontinuous in the tangential direction because the surface is assumed to be frictionless. This paper will not address the case of friction. The Riemann force in the normal direction, \( \mathbf{F}_{n} \cdot \mathbf{n}_{CS} = \mathbf{N} \cdot \sigma_{p} \cdot \mathbf{n}_{CS} \), is equal to the mechanical force plus a viscous force. The left-hand side of Eq. (13) is a viscous force per area acting on an iota face in the contact normal direction. The normal force from all the iota faces directly connected to the penetrating node (Fig. 5) is

![Fig. 4. The nodal control volume used for the Riemann-like problem along a contact surface is shown with a dashed line. For illustrative purposes, the contact surfaces are pulled apart. The control volume above corresponds to the example shown in Fig. 3.](image-url)
Fig. 5. The control volume above is the bottom portion of the control volume shown in Fig. 4. The iota surfaces connected to the penetrating node, \( i \in P_{\text{pen}} \), are denoted with a bold black line.

\[
\sum_{i \in P_{\text{pen}}} F_i^r \cdot n_{\text{CS}} n_{\text{CS}} = \sum_{i \in P_{\text{pen}}} \left( \mu_{c(i)} (u_p^r - u_{c(i)}) |a_{c(i)}| \cdot N_i \cdot n_{\text{CS}} n_{\text{CS}} + N_i \cdot \sigma_{c(i)} \cdot n_{\text{CS}} n_{\text{CS}} \right),
\]

where the summation over all the iota faces connected to the penetrating node is denoted by \( i \in P_{\text{pen}} \). These iota surfaces are denoted in Fig. 5 with a bold black line. Note, Eq. (14) only accounts for force contributions from the penetrating node iota surfaces. The derivation in this section corresponds to a node that is penetrating or touching a contact surface so Eq. (14) will be modified to include normal force contributions from the contacting node set. The steps below discuss how the normal Riemann forces from the contacting node set are included in the calculation.

The force from the contacting node set is found via a Riemann jump equation on the iota faces in the nodal control volume on the contact side (Fig. 6). The Riemann equation for an iota face of one contacting node is

\[
\mu_{c(i)} (u_p^r - u_{c(i)}) |a_{c(i)}| \cdot N_i \cdot \phi_{\text{pen}} n_{\text{CS}} n_{\text{CS}} = N_i \cdot (\sigma_{c(i)} - \sigma_{c(i)}) \cdot \phi_{\text{pen}} n_{\text{CS}} n_{\text{CS}}.
\]

The Riemann equation above uses an area fraction, \( \phi \), on the contacting node iota surface areas. The area fraction is used to account for misalignment between the contacting node surface and the penetrating node surface. The area fraction is the same for all iota faces connected to the contacting node. If the mesh is perfectly aligned over the control volume region, then all the surface area fractions are equal to 1.0 and the Riemann equation is identical on all iota surfaces – penetrating side and contacting side respectively. Section 3.5 provides additional details on the surface area fractions. The normal Riemann force contribution by a single contacting node is

\[
\sum_{i \in P_{\text{pen}}} F_i^r \cdot n_{\text{CS}} n_{\text{CS}} = \phi_{\text{pen}} \sum_{i \in P_{\text{pen}}} \left( N_i \cdot \sigma_{c(i)} \cdot n_{\text{CS}} n_{\text{CS}} + \mu_{c(i)} (u_p^r - u_{c(i)}) |a_{c(i)}| \cdot N_i \cdot n_{\text{CS}} n_{\text{CS}} \right).
\]

The force from the contacting node set on the penetrating node includes both mechanical and viscous forces. If the area fraction is equal to 1.0, then the normal force equation for a contacting node (Eq. (16)) is identical to the normal Riemann force equation of the penetrating node (Eq. (15)). Next, the normal force from all the contacting nodes is merely a sum over all the contacting node set. Therefore, the force from the contacting node set on the penetrating node is

\[
\sum_{p_{\text{con}} \in \text{CNS}} \sum_{i \in P_{\text{pen}}} F_i^r \cdot n_{\text{CS}} n_{\text{CS}} = \sum_{p_{\text{con}} \in \text{CNS}} \left( \phi_{\text{pen}} \sum_{i \in P_{\text{pen}}} \left( N_i \cdot \sigma_{c(i)} \cdot n_{\text{CS}} n_{\text{CS}} + \mu_{c(i)} (u_p^r - u_{c(i)}) |a_{c(i)}| \cdot N_i \cdot n_{\text{CS}} n_{\text{CS}} \right) \right),
\]

where CNS means the contacting node set. The expression above is the sum of the area-weighted normal Riemann forces (i.e. both mechanical and viscous parts) for each contacting node involved in the contact calculation. It is important to note that the expression above assumes the normal Riemann velocity component, \( u_p^r \cdot n_{\text{CS}} n_{\text{CS}} \), is the same for all nodes involved in the contact calculation. Small errors may arise on unaligned meshes from assuming the same normal Riemann velocity component for all the nodes in the contact calculation. This numerical error is a function of the mesh misalignment and the mesh size. As will be shown in the test problem section, this error is small, even on coarse meshes.

Conservation of momentum requires the normal component of the Riemann forces acting on the node to sum to zero.

\[
\sum_{i \in P_{\text{pen}}} F_i^r \cdot n_{\text{CS}} n_{\text{CS}} + \sum_{p_{\text{con}} \in \text{CNS}} \sum_{i \in P_{\text{pen}}} F_i^r \cdot n_{\text{CS}} n_{\text{CS}} = 0.
\]

Substituting Eqs. (14) and (17) into the conservation of the momentum Eq. (18) produces an equation for the penetrating nodal Riemann velocity in the normal direction, \( u_p^r \cdot n_{\text{CS}} n_{\text{CS}} \). Solving for the normal Riemann velocity component of the penetrating node gives

Fig. 6. The control volume above is the top portion of the control volume shown in Fig. 4. The iota surfaces in the contacting node set, \( i \in \text{CNS} \), are denoted with a bold black line.
\[ \mathbf{u}_p \cdot \mathbf{n}_{CS} = \frac{\sum_{i \in \text{pen}} (\mu_{c(i)} \mathbf{u}_{c(i)} \cdot \mathbf{n}_{c(i)} - \mathbf{N}_i \cdot \mathbf{n}_{CS} - \mathbf{N}_i \cdot \mathbf{n}_{CS}) + B'}{\sum_{i \in \text{pen}} (\mu_{c(i)} \mathbf{a}_{c(i)} \cdot \mathbf{n}_i) + A} \]  

where

\[ A' = \sum_{i \in \text{pen}} \sum_{i \in \text{pen}} (\phi_{pen} \sum_{i \in \text{pen}} (\mu_{c(i)} \mathbf{a}_{c(i)} \cdot \mathbf{n}_i)), \]  

\[ B' = \sum_{i \in \text{pen}} (\phi_{pen} \sum_{i \in \text{pen}} (\mu_{c(i)} \mathbf{a}_{c(i)} \cdot \mathbf{n}_i \mathbf{u}_{c(i)} \cdot \mathbf{n}_{CS} - \mathbf{N}_i \cdot \mathbf{a}_{c(i)} \cdot \mathbf{n}_{CS})). \]

The \( A' \) and \( B' \) terms contain the contributions from the contacting node set. A free surface solution is achieved by setting \( A' \) and \( B' \) equal to zero. \( A' \) and \( B' \) are equal to zero when the surfaces are not in contact or when their inclusion would result in tension. The contact surface will be in tension if the application of \( A' \) and \( B' \) in Eq. (19) results in a force pointed in the outward normal direction.

The solution process for the nodal Riemann velocity in the contact surface normal direction of a penetrating node is as follows: (1) sum over all the iota faces connected to a contacting node, (2) multiply by an area fraction, (3) sum over all the contacting nodes influencing the penetrating node, and (4) calculate the nodal Riemann velocity in the normal direction using Eq. (19). The solution process is applied to every penetrating node. The force in the normal direction on each iota face connected to the penetrating node, \( F_i \mathbf{n}_{CS} = N_i \cdot \mathbf{n}_{CS} \), is calculated by substituting the penetrating nodal Riemann velocity into the Riemann equation for the iota face (Eq. (13)). The Riemann force and the Riemann velocity in the contact surface normal direction are used in the conservation of momentum and total energy equations for the zone (Eqs. (2) and (3)). The nodal solver is valid for an arbitrary number of iota faces, so the contact surface solution approach above will work with arbitrary polyhedra in both two and three dimensions. For simplicity purposes, we restricted the diagrams to simple mesh topologies in two-dimensions.

The Riemann-like problem above includes velocity and stress information from both contact surfaces. The nodes on both sides of the contact surfaces will move at the same Riemann velocity on perfectly aligned Lagrangian meshes, or unaligned meshes with a one-dimensional flow. The same Riemann velocity on both surfaces ensures no interpenetration of the Lagrangian meshes. However, the calculated Riemann velocity on both sides of the contact can be slightly different on unaligned meshes with two-dimensional flows. The small difference in the Riemann velocity will allow the two contact surfaces to interpenetrate. In some instances, it may take a few cycles for the nodes on each side of the contact surface to have the same Riemann velocity. The proposed contact algorithm does not include a correction to remove a penetration error so interpenetration errors will persist throughout the calculation. As seen in the test problems, the interpenetration errors are extremely small even on unaligned meshes with two-dimensional flows.

The interpenetration errors on two-dimensional flows with unaligned meshes arise from assuming the same normal Riemann velocity component for every node inside the nodal control volume. Specifically, momentum conservation was enforced (Eq. (18)) assuming the nodal Riemann velocities were all equal. Such an assumption is only true in a one-dimensional Cartesian flow or a purely radial flow on a polar mesh where the mesh is aligned across the contact surface. The later case corresponds to the limiting case discussed below in Section 3.3. If the nodal Riemann velocities are indeed different, then a momentum conservation error will be generated in the calculation. The momentum conservation error will in turn produce a conservation of total energy error. The conservation errors are a function of the velocity variation along the contact surface, the mesh resolution, and the mesh misalignment. As seen in the test problems, the solutions to radial flow problems (Sections 4.2 and 4.3) with an unaligned meshes are in good agreement with the calculations on aligned meshes, which indicates the conservation errors can be small.

3.3. A limiting case

The contact surface approach described in this paper will produce the same normal nodal Riemann velocity and normal Riemann force as a continuous mesh (a mesh without a contact surface) with first-order when the mesh is aligned across the contact surface. This limiting case is important, because the normal nodal Riemann velocity and normal Riemann force should be identical with or without a contact surface. In addition, the limiting case ensures the contact surface method preserves symmetry on radial flow problems with an equal angle polar mesh, because the nodal solution on a continuous mesh will preserve symmetry on equal angle polar meshes [3].

To see that the limiting case is satisfied, consider the case of an aligned mesh across the contact surface. The normal Riemann velocity component (Eq. (19)) for an aligned mesh becomes

\[ \mathbf{u}_p \cdot \mathbf{n}_{CS} = \frac{\sum_{i \in \text{pen}} (\mu_{c(i)} \mathbf{u}_{c(i)} \cdot \mathbf{n}_{c(i)} - \mathbf{N}_i \cdot \mathbf{a}_{c(i)})}{\sum_{i \in \text{pen}} (\mu_{c(i)} \mathbf{a}_{c(i)} \cdot \mathbf{n}_i)} \cdot \mathbf{n}_{CS}. \]

The area fractions for an aligned mesh are 1.0. The contact surface normal vector is the same for all quantities so it was factored outside the summation. The summation is over all the iota faces on the contact side and the penetrating side, hence
i \in p_{pen}\cap p_{con}$. The iota faces in the summation in Eq. (22) are the same iota faces for a continuous mesh. The expression inside the brackets is identical to the nodal solver used with a continuous mesh (Eq. (8)). To see this, the velocity and stress in Eq. (8) are decomposed into the normal and tangential components of the contact surface. Specifically, the velocity is $\mathbf{u} = \mathbf{u} \cdot \mathbf{n}_{CS} + \mathbf{u} \cdot \mathbf{t}_{CS}$ and the stress is $\mathbf{\sigma} = \mathbf{\sigma} \cdot \mathbf{n}_{CS} + \mathbf{\sigma} \cdot \mathbf{t}_{CS}$. The component normal to the contact surface is identical to Eq. (22).

The Riemann force in the normal direction, $\mathbf{F} \cdot \mathbf{n}_{CS} = \mathbf{N} \cdot \mathbf{n}_{CS}$, is calculated by evaluating the Riemann equation for an iota face (Eq. (13)) with the normal Riemann velocity component. The normal Riemann velocity component is the same for both cases (aligned mesh with a contact surface and a continuous mesh) so the resulting Riemann force in the normal direction is the same with both mesh topologies.

The limiting case is satisfied if the same inputs are used in the respective nodal solver. Currently, the limiting case is only satisfied with the first-order accurate approach because the contact surface approach neglects the gradient in the projection step. The limiting case could be satisfied with the second-order accurate approach if the same gradient is used in the projection step with a contact surface and without the contact surface.

### 3.4. Tangential direction calculation

The above discussion addressed the normal Riemann velocity component and the corresponding normal force. The tangential Riemann velocity component and the associated tangential forces are discussed in this section. The algorithm presented in this paper corresponds to a frictionless surface. The tangential velocity component and tangential force acting on the penetrating node does not include contributions from the contacting node set; as a result, the tangential Riemann velocity component corresponds to a free surface solution. A Riemann-like problem is solved at the penetrating node. The control volume for the tangential direction will only include the iota faces connected to the penetrating node. The Riemann problem on each iota face directly connected to the penetrating node (Fig. 5) is

$$
\mathbf{\mu}_{ci}(\mathbf{u}_i^p - \mathbf{u}_i) \cdot \mathbf{a}_{ci} \cdot \mathbf{n}_i \cdot \mathbf{t}_{CS} = \mathbf{n}_i \cdot (\mathbf{\sigma}_{ci} - \mathbf{\sigma}_{ci}) \cdot \mathbf{t}_{CS}.
$$

A dot product is used to select the component in the tangential direction, $\mathbf{t}_{CS}$, of the contact surface at the penetrating node. The $\mathbf{t}_{CS}$ unit vector is defined in Section 3.6. All the tangential forces acting on the penetrating node is

$$
\sum_{i \in p_{pen}} (\mathbf{F}_i \cdot \mathbf{t}_{CS}) = \sum_{i \in p_{pen}} (\mathbf{\mu}_{ci}(\mathbf{u}_i^p - \mathbf{u}_i) \cdot \mathbf{a}_{ci} \cdot \mathbf{n}_i \cdot \mathbf{t}_{CS} + \mathbf{N}_i \cdot \mathbf{\sigma}_{ci} \cdot \mathbf{t}_{CS}).
$$

The conservation of momentum equation requires the sum of all tangential forces acting on the penetrating node be equal to zero. Since there is no friction on the contact surface, conservation of momentum requires Eq. (24) be equal to zero. Solving for the tangential velocity of the penetrating node gives

$$
\mathbf{u}_i^p \cdot \mathbf{t}_{CS} = \frac{\sum_{i \in p_{pen}} (\mathbf{\mu}_{ci} \cdot \mathbf{a}_{ci} \cdot \mathbf{n}_i \cdot \mathbf{u}_i) \cdot \mathbf{t}_{CS} - \mathbf{N}_i \cdot \mathbf{\sigma}_{ci} \cdot \mathbf{t}_{CS})}{\sum_{i \in p_{pen}} \mathbf{\mu}_{ci} \cdot \mathbf{a}_{ci} \cdot \mathbf{n}_i}.
$$

The tangential Riemann force on an iota face connected to the penetrating node, $\mathbf{F}_i \cdot \mathbf{t}_{CS} = \mathbf{N}_i \cdot \mathbf{\sigma}_{ci} \cdot \mathbf{t}_{CS}$, is found by substituting the tangential velocity into Eq. (23) and multiplying by the iota surface area, $A_i$. The nodal Riemann tangential velocity component and associated Riemann force corresponds to a free surface solution. The tangential Riemann velocity component and corresponding tangential Riemann force are used in the conservation of momentum and total energy equations for the zone (Eqs. (2) and (3)).

### 3.5. Surface area fractions

Up to this point, little has been said on how to calculate the surface area fraction weights, $\phi_{pen}$. The surface area fraction weights must account for the surface misalignment between a contacting node and a penetrating node. The control volume of a contacting node may extend beyond the penetrating node as illustrated in Fig. 4. Each node in the contacting node set has an area, which is denoted as $A_{pen}$. The overlapping area of the contacting node on the penetrating node surface is $A_{pen} \cap A_{pen}$. The penetrating node surface defines the extent of the contact surface calculation, so if a contacting node does not overlap the penetrating node then $\phi_{pen} = 0$. The area fractions are equal to the ratio of the overlapping area to the contacting node surface area,

$$
\phi_{pen} = \frac{A_{pen} \cap A_{pen}}{A_{pen}}.
$$

For example, if only half of the contacting node surface area projects onto the penetrating surface, then the area fraction is 0.5. If the entire surface projects onto the penetrating surface, then the area fraction is 1.0. The surface area fraction calculation is illustrated in Fig. 7.
3.6 Contact surface vectors

The contact surface unit normal vector, \( \mathbf{n}_{CS} \), at the penetrating node is equal to the surface area weighted average of the two unit normal vectors connected to the penetrating node (Fig. 8). The area weights are \( \frac{A_{pen}}{A_{pen} + A_{appen}} \), where the overlapping area, \( A_{pen} \cap A_{appen} \), is denoted with a bold black arrow.

4. Test problems

Multiple tests were performed to assess the accuracy of the new contact surface approach. The tests are:

1. A slab impact problem [8]
2. The Noh problem [29]
3. The Sedov blast wave problem [30]
Fig. 9. The one-dimensional slab impact problem is shown above at several different times. The slabs are initially 2 cm long and 0.2 cm tall. The slabs are initially separated by 0.18 cm. The slabs collide at 1 $\mu$s. The aluminum is the left slab and the stainless steel is the right slab. The mesh above is unaligned along the contact surface. The aluminum mesh uses 11 nodes and the stainless steel mesh uses 12 nodes in the vertical direction. The contact algorithm allows the slabs to impact and separate.

Fig. 10. The slab impact test problem pressure and density results are shown above corresponding to a time of 2.25 $\mu$s after the collision. The collision occurred at 2.18 cm. The calculations on the left side use a first-order accurate solution approach for every node in the domain. The calculations on the right side use a second-order solution approach for the internal nodes and a first-order accurate solution for the nodes on the contact surface. The contact surface calculations agree well with the analytic solution.

4. Two Taylor-Anvil impact experiments [32,13]
5. Two rod–rod impact problems
6. A flyer plate experiment [19]
7. Three exploding aluminum shell experiments [38].
The Noh and Sedov test problems have a contact surface placed in the middle of the mesh. The slab impact, Noh, and Sedov blast wave problems are performed with aligned and unaligned meshes at the contact surface. The goal of using an unaligned mesh at the contact surface in these test problems is to assess the ability of the contact surface method to preserve symmetry. The next series of test problems are experiments. The first experiments calculated are Taylor-Anvil experiments, which involve a collision between a metal cylinder and a plate. The goal of the Taylor-Anvil calculations is to test the ability of the contact surface algorithm to capture collision and the subsequent tangential slip along the material-material interface. The Taylor-Anvil calculations start with an unaligned mesh and the mesh deforms tangentially along the contact surface. The next tests are rod–rod collisions. The rods are identical to the rods used in the Taylor-Anvil experiments. In this test problem, the rod–rod collision results are compared to calculations with a single rod that uses a reflected boundary condition. The goal of the rod–rod calculations is to (1) verify the symmetry of the deformation on both sides of the contact surface, and (2) confirm the contact surface approach produces similar deformation as a reflective boundary condition. The next experiment is a flyer plate experiment that involves a copper plate impacting a copper target. The flyer plate experiment will test the accuracy of contact surface approach at generating the correct shock structure. The last set of experiments calculated are exploding aluminum shells. These experiments used different sized gaps between the explosive charge and the aluminum shell. The exploding shell experiments will test the ability of the algorithm to capture collision and the subsequent push caused by the expanding gaseous products. The test problems above have an analytic solution or experimental data so it is possible to quantify the accuracy of the new contact surface method.

4.1. Slab impact problem

Cooper [8] provides a one-dimensional solution for two slabs of material colliding. The test problem uses a linear equation of state with the Gruneisen gamma equal to zero [39]. The details of the linear equation of state are provided in
Appendix A in [3]. The impact test problem involves a block of aluminum 2024 traveling at 0.18 cm/μs colliding with a stationary block of 304 stainless steel. In this work, each slab is 2 cm long and 0.2 cm tall. The slabs are initially separated. A

Fig. 12. The 1D Cartesian Noh results are shown above corresponding to a time of 0.6 μs after the collision. The calculations on the left side use a first-order accurate solution approach for every node in the domain. The calculations on the right side use a second-order solution approach for the internal nodes and a first-order accurate solution for the nodes on the contact surface. The calculations capture the pressure and density given by the analytic solution away from the contact surface. Wall heat errors [29] are present in both calculations at the contact surface as expected.

Fig. 13. The unaligned XY Noh mesh is shown above.
0.18 cm gap is placed between the two slabs of material. The slabs will impact 1 μs after the start of the calculation. The reference density of aluminum 2024 is 2.785 g/cc and the linear equation of state parameters are $C_0 = 0.5328 \text{ cm/μs}$ and $s = 1.338$. The reference density of the stainless steel is 7.896 g/cc and the linear equation of state parameters are $C_0 = 0.4569 \text{ cm/μs}$ and $s = 1.490$. The shock pressure and particle velocity after the collision are 0.24 Mbar and 0.0563 cm/μs respectively.

The mesh resolution used with the impact test problem is 0.02 cm, which corresponds to 11 nodes in the vertical direction and 101 nodes in the horizontal direction. Calculations were performed with an aligned and an unaligned mesh. The

![Image](image.png)

**Fig. 14.** The XY Noh results are shown above corresponding to a time of 0.6 μs. The bottom image shows a color plot of the pressure field [Mbar] corresponding to a second-order solution for the interior nodes on an unaligned mesh. The calculations on the left side use a first-order accurate solution approach for every node in the domain. The calculations on the right side use a second-order solution approach for the internal nodes and a first-order accurate solution for the nodes on the contact surface. The first-order solution everywhere was not as accurate as using a second-order solution for the internal nodes. The second-order solution matches the density and pressure given by the analytic solution away from the wall. As is common with the Noh problem, wall heat errors are present in both calculations. (For interpretation of the references to color in this figure legend, the reader is referred to the web version of this article.)
unaligned mesh used one more zone vertically (contact tangential direction) in the stainless steel (i.e. 12 nodes). Reflected boundary conditions were located along the top and bottom edges of the slabs, because the problem is one-dimensional. The solution with an aligned and unaligned mesh should be one-dimensional. Fig. 9 shows the collision problem, where the slab on the left is aluminum and the slab on the right is stainless steel. The collision sequence shown in Fig. 9 uses an unaligned mesh. Fig. 10 plots the pressure and density and Fig. 11 plots the internal energy and the particle velocity. Both of these figures provide results for the aligned and unaligned mesh cases. The CCH calculation with the new contact surface approach agrees well with the analytic solution. There is, however, an internal energy error at the contact surface. The internal energy error is the well known Noh wall heat error[29]. The internal energy error only occurs in a few zones near the contact surface with the second-order calculation, whereas the wall heat errors propagate several millimeters away from the contact surface with a first-order accurate solution. The solution is the same on an aligned and unaligned mesh, because the test problem is a one-dimensional Cartesian problem. The contact surface allowed the slabs to impact and then separate. The separation can be seen in the bottom image in Fig. 9. The separation is achieved by not mapping the tensile wave across the contact surface.

4.2. Noh with contact surface

The one-dimensional Cartesian Noh problem is equivalent to two slabs of gas colliding where the initial velocity magnitude of each slab is 1 cm/μs. The initial density is 1 g/cc. In this work, gamma is equal to 5/3. The gas slabs are initially 1.0 cm long and 0.2 cm and tall. A 2 cm gap is placed between the slabs. The slabs will collide 1 μs after the start of the calculation. The shock pressure and density after the collision are 1.33 Mbar and 4.0 g/cc respectively. The mesh resolution used with the planar Cartesian Noh problem is 0.02 cm. Calculations were performed with an aligned and an unaligned mesh. The unaligned mesh has one additional zone vertically in one of the slabs (i.e. 12 nodes). The zoning normal to the contact surface is not changed in the unaligned mesh case (i.e. 0.02 cm). Fig. 12 plots the pressure and density for the aligned and unaligned mesh cases. The calculated pressure agrees well with the analytic solution; however, density errors are present near the wall. The density errors near the wall are a well known issue in hydrodynamics[29]. A noticeable overshoot in density is present near the wall in the second-order accurate calculation. The overshoot is attributed to switching from a second-order solution to a first-order solution at the contact surface (i.e. gradients are zero in the zones along a contact surface or a free surface). Future work will focus on second-order solutions at a contact surface and a free surface.

The next test problem is the two-dimensional XY Noh problem. The XY Noh problem involves a gamma-law gas instantaneously stopping at the origin, where the initial velocity is −1 cm/μs. In the XY Noh problem, the velocity vector at each grid location is pointing at the origin. The analytic solution for the pressure and the density are 5.33 Mbar and 16 g/cc respectively. The contact surface is located at a radius of 0.5 cm. The aligned mesh case will use 12 zones in the angular direction. The unaligned mesh case will use 12 angular zones in the inner mesh and 13 angular zones in the outer mesh. The angular resolution of both meshes (aligned and unaligned) was reduced to 6 zones in the first two radial zones to improve the aspect ratio of the mesh at the origin. Coarsening the mesh created 5-noded elements. The same Riemann-like nodal solver was used at every node in the 5-noded elements. The details of the 5-noded elements are more easily seen in the Sedov problem (Fig. 16). The unaligned mesh used in the Noh problem is shown in Fig. 13.
Fig. 16. The Sedov results are shown above corresponding to a time of 1 µs. The initial location of the contact surface is 0.5 cm. The bottom image shows a color plot of the pressure field [Mbar] corresponding to a second-order solution for the interior nodes on an unaligned mesh. The calculations on the left side use a first-order accurate solution approach for every node in the domain. The calculations on the right side use a second-order solution approach for the internal nodes and a first-order accurate solution for the nodes on the contact surface. The mesh was coarsened in the angular direction near the origin to improve the aspect ratio of the zones. The coarsening created zones with 5 nodes. The Riemann-like problem was solved at every node in the problem including the nodes along the mesh coarsening boundary. As illustrated above, the Riemann-like problem is stable with the zones that have 5 nodes. (For interpretation of the references to color in this figure legend, the reader is referred to the web version of this article.)

Table 1
The initial and final dimensions of the copper rod in the Taylor-Anvil experiment are shown below.

<table>
<thead>
<tr>
<th>Velocity (cm/µs)</th>
<th>Initial diameter (cm)</th>
<th>Initial length (cm)</th>
<th>Final diameter (cm)</th>
<th>Final length (cm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0083</td>
<td>1.91</td>
<td>7.51</td>
<td>2.36</td>
<td>7.03</td>
</tr>
<tr>
<td>0.0205</td>
<td>1.89</td>
<td>7.50</td>
<td>3.66</td>
<td>5.45</td>
</tr>
</tbody>
</table>
Table 2
The calculated final dimensions of the copper rod in the Taylor-Anvil are shown below along with the associated error. The rod dimensions were measured 149 μs after the collision.

<table>
<thead>
<tr>
<th>Velocity (cm/μs)</th>
<th>Final diameter (cm)</th>
<th>Final length (cm)</th>
<th>Diameter error (cm)</th>
<th>Length error (cm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0083</td>
<td>2.25</td>
<td>7.13</td>
<td>−0.11</td>
<td>0.10</td>
</tr>
<tr>
<td>0.0205</td>
<td>3.37</td>
<td>5.82</td>
<td>−0.29</td>
<td>0.37</td>
</tr>
</tbody>
</table>

Fig. 17. Two Taylor-Anvil experiments were used to test the contact surface algorithm on a problem that has material strength and tangential flow along the contact surface. The horizontal axis is the radial direction. The materials in the experiment are copper (blue), steel (gray), and aluminum (green). A small gap separates the steel plate from the copper rod at the beginning of the calculation. The steel plate collides with the copper rod 1 μs after the start of the calculation. Both images correspond to a time of 149 μs after the collision. The calculations are within 4.7% percent of the experimental data for the impact at 0.0083 cm/μs and within 8.0% for the impact at 0.0205 cm/μs. The increased error at higher velocities might be the result of treating the contact surface as first-order. (For interpretation of the references to color in this figure legend, the reader is referred to the web version of this article.)

Table 3
The calculated final dimensions of the rod–rod collision problem are shown below. The rod dimensions were measured 149 μs after the collision in every calculation. The rod impact problem was also calculated using a reflected boundary condition calculations (i.e. no contact surface). The results using a reflected boundary condition (BC) are shown in the third and forth row. The rod–rod calculations are in excellent agreement with the reflected boundary condition calculations. Slight differences should be expected since the reflected boundary condition will support tension, whereas, the contact surface does not support tension.

<table>
<thead>
<tr>
<th>Calculation</th>
<th>Velocity (cm/μs)</th>
<th>Final diameter (cm)</th>
<th>Final length (cm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rod–rod</td>
<td>0.0083</td>
<td>2.32</td>
<td>7.06</td>
</tr>
<tr>
<td>Rod–rod</td>
<td>0.0205</td>
<td>3.70</td>
<td>5.44</td>
</tr>
<tr>
<td>Reflected BC</td>
<td>0.0083</td>
<td>2.38</td>
<td>7.06</td>
</tr>
<tr>
<td>Reflected BC</td>
<td>0.0205</td>
<td>3.75</td>
<td>5.43</td>
</tr>
</tbody>
</table>
The XY Noh results are shown in Fig. 14 corresponding to 0.6 $l_s$. The errors near the wall are more substantial than the planar Cartesian Noh problem, which is consistent with the results of [29]. The contact surface generates an error in the density field similar to the density error at the wall. The density perturbation at the contact is more pronounced in the second-order accurate calculation. The pressure and density fields with the second-order accurate solution are in good agreement with the analytic solution away from the wall. The calculations with an unaligned mesh along the contact surface performed very well. The unaligned mesh results are very similar to the results with an aligned mesh. Lastly, a comparison is provided in Fig. 15 between a calculation with a contact surface and a continuous mesh using the first-order accurate approach at every node. The solution with a contact surface is the same as the solution with a continuous mesh, which agrees with the derivation in Section 3.3.

4.3. Sedov with contact surface

Sedov [30] obtained an analytic solution for a blast wave in a gamma-law gas that is generated by an energy source at a point. The Sedov problem is modeled using a finite energy source at the origin of a polar mesh. The Lagrangian polar mesh is 1.2 cm in radius with 120 zones. The Sedov problem uses the same resolution as the XY Noh test problem. The angular mesh resolution for the aligned and unaligned mesh cases are identical to the XY Noh problem. The unaligned Sedov mesh has 12 angular zones in the inner mesh and 13 angular zones in the outer mesh (Fig. 13); likewise, the angular resolution is reduced.
to 6 zones near the origin to improve the aspect ratio of those zones. The contact surface is placed at a radius of 0.5 cm. The internal energy in Sedov was initialized such that the shock position is located at 1 cm at 1 μs. The initial density is 1 g/cc and the initial source specific internal energy is 448.91 Mbar cm³/g. The gamma is 5/3.

The Sedov results are shown in Fig. 16. The second-order accurate calculation in the interior nodes generated an error in the density field along the contact surface. The contact surface density error is similar to the density error observed in the XY Noh problem. Using a first-order solution along the contact creates a dissipation error in the solution that manifests as a density drop and spike. Next, the Sedov results with an unaligned mesh agree well with the results using an aligned mesh.

4.4. Taylor-Anvil experiments

The Taylor-Anvil experiment [32] uses the collision between a cylinder of test material and a target plate to study high-strain rate behavior of the material. Our goal of the modeling a Taylor-Anvil experiment is to test the ability of the contact surface method to capture tangential slip along the contact surface.

The experiments modeled in this work have a steel plate traveling at 0.0083 cm/μs or 0.0205 cm/μs colliding with a rod [13]. The plate is hardened maraging 300 steel. The steel plate is 0.6 cm thick and it is attached to a 17.8 cm thick aluminum carrier sabot. The projectile (plate and sabot) is 7.62 cm in diameter. The rod is oxygen-free electronic copper. The nominal diameter and nominal length of the rod is 1.9 cm and 7.5 cm respectively. The initial and final dimensions of the copper rod in the experiment are shown in Table 1.

The Taylor-Anvil calculation was initialized with a small gap between the plate and the rod. The materials collide 1 μs after the start of the calculation, so the gap size is a function of the initial velocity. The initial mesh resolution used with this test problem is 0.1 cm nominally. The aluminum sabot uses a sesame tabular equation of state, 3717, [23] and a kospall strength model [34]. The copper rod and steel plate use a linear equation of state with a Gruneisen coefficient to capture the off Hugoniot states. The copper equation of state properties are: \( q_0 = 8.930 \text{ g/cc} \), \( C_0 = 0.394 \), \( s = 1.49 \), and \( \Gamma = 2.0 \). The steel equation of state properties are: \( q_0 = 8.0 \text{ g/cc} \), \( C_0 = 0.4578 \), \( s = 1.33 \), and \( \Gamma = 1.67 \). The copper rod and steel plate use a strength model where the flow stress is a linear function of the equivalent plastic strain. The details of the strength model are provided in Appendix A of [3]. The copper strength properties are: yield stress \( Y_0 = 0.004 \text{ Mbar} \), hardening modulus \( Y_H = 0.001 \), and shear modulus \( G = 0.4333 \). The steel strength properties are: yield stress \( Y_0 = 0.02 \text{ Mbar} \), hardening modulus \( Y_H = 0.0 \), and shear modulus \( G = 0.801 \). The steel does not include a hardening term.

The results from the Taylor-Anvil calculations are shown in Table 2, and images of the calculations are shown in Fig. 17. The calculation with an initial velocity of 0.0083 cm/μs was within a few percent of the experimental data. Specifically, the relative percent error in the final rod diameter at the contact surface is –4.7% and the relative error in the final rod length is 1.4%. The negative sign indicates the calculated diameter was less than the diameter in the experiment. The calculation with an initial velocity of 0.0205 cm/μs was in reasonable agreement with experimental data. The relative percent error in the final rod diameter at the contact surface is –8.0% and the relative error in the final rod length is 6.8%. The error increased with the larger initial velocity, which might be the result of treating the contact surface as first-order. The important point is the contact surface allowed the copper rod to deform radially and longitudinally in a manner consistent with the experiment.

The cell-centered pressure in the rod–rod impact problem is shown above 6 μs after the collision. The pressure range is different for each initial velocity and has units of Mbar. The images above illustrate the contact surface will allow the two rods separate when the contact surface goes into tension. The separation shown above is caused by an unloading wave from the free surface. The rod–rod separation above is most noticeable with an initial velocity of 0.0205 cm/μs. In contrast, the rod–rod separation above is very small with an initial velocity of 0.0083 cm/μs.

Fig. 19. The cell-centered pressure in the rod–rod impact problem is shown above 6 μs after the collision. The pressure range is different for each initial velocity and has units of Mbar. The images above illustrate the contact surface will allow the two rods separate when the contact surface goes into tension. The separation shown above is caused by an unloading wave from the free surface. The rod–rod separation above is most noticeable with an initial velocity of 0.0205 cm/μs. In contrast, the rod–rod separation above is very small with an initial velocity of 0.0083 cm/μs.
4.5. Rod–rod impact test

The rod–rod impact problems have two rods with equal and opposite velocities that impact each other. For comparison purposes, the rod–rod impact problem is also calculated using a reflected boundary condition. The rod–rod impact results should be nearly identical to the results obtained with a reflected boundary condition. Small differences should be expected because the reflected boundary condition supports tension, whereas, the contact surface does not support tension. These impact tests use the same velocity magnitudes and initial rod dimensions as the Taylor-Anvil experiment above. The initial velocities are 0.0083 cm/µs and 0.0205 cm/µs.

The rod–rod impact tests use the same mesh resolution, the same copper equation of state, and the same copper strength model as the Taylor-Anvil calculations above. The results for these calculations are provided in Table 3 and Figs. 18–20. The rod–rod impacts produce results very similar to the single rod calculations that use a reflected boundary condition. The deformation in the top rod is a mirror image of the deformation in the bottom rod, which illustrates the contact algorithm will preserve symmetry across the contact. Furthermore, the images in Figs. 19 and 20 show symmetrical separation between the two colliding rods. The rod–rod impact results produce deformation very similar to the Taylor-Anvil experiments calculated above.

(a) Initial velocity is 0.0083cm/µs  (b) Initial velocity is 0.0205cm/µs

Fig. 20. The rod–rod impact calculation was run until 300 µs after the collision to demonstrate the algorithm is capable of allowing the Lagrangian meshes to separate. As shown above, significant separation has occurred.
4.6. Flyer plate experiment

The flyer plate experiment involves a 0.6 mm copper plate impacting a 1.6 mm copper target that is backed by a thick layer of PMMA plastic (2.4 mm) [19]. The flyer plate has an initial velocity of 0.016 cm/µs, which generates a 0.029 Mbar shock in the target. The PMMA plastic contains a Manganin gage 0.5 mm from the copper-PMMA interface. This experiment generates a spall plane in the copper target when the unloading waves from the free surfaces collide. The experiment is shown in Fig. 21. The copper equation of state and strength models are identical as those used in the Taylor Anvil experiment.

Calculations of a flyer plate experiment were performed. The flyer plate experiment consists of a copper flyer colliding with a copper target. A piece of PMMA plastic is behind the target and it contains a Manganin gauge. The copper material is blue and the PMMA is orange. A gap is placed between the flyer plate and the target to test the ability of the contact surface to handle a collision. (For interpretation of the references to color in this figure legend, the reader is referred to the web version of this article.)

![Fig. 21](image1.png)

Fig. 21. Calculations of a flyer plate experiment were performed. The flyer plate experiment consists of a copper flyer colliding with a copper target. A piece of PMMA plastic is behind the target and it contains a Manganin gauge. The copper material is blue and the PMMA is orange. A gap is placed between the flyer plate and the target to test the ability of the contact surface to handle a collision. (For interpretation of the references to color in this figure legend, the reader is referred to the web version of this article.)

![Fig. 22](image2.png)

Fig. 22. The cell-centered pressure versus time history at the Manganin gauge for the flyer plate experiment is shown above. The time in the plot corresponds to the start of the calculation. There is a 1 µs delay between the start of the calculation and the collision. The goal here is not to validate a particular failure model, but rather, test the accuracy of the contact surface algorithm. As shown above, the contact surface approach produces a pressure response that is in good agreement with the experimental data up to the point where the spall signal arrives at the gauge. A failure model such as TEPLA [20,2] is required to capture the entire pressure response in this experiment. The time when the spall signal arrives at the gauge is shown in the plot with a dashed line.
experiments above. Our focus here is on validating the contact surface approach, and not a particular failure model; as a result, we seek to match the experimental data up to the point where the spall signal arrives at the gauge. The reference density of PMMA is 1.184 g/cc and the Gruneisen equation of state parameters are $C_0 = 0.2598 \text{ cm/μs}$, $s = 1.516$, and $\Gamma = 0.97$. A small gap is placed between the flyer plate and the target. The flyer plate will collide with the target 1 μs after the start of the calculation.

The calculated pressure response is compared to the pressure data from the Manganin gauge in Fig. 22. The goal here is to correctly capture the pressure versus time history up to the point when the spall signal arrives. As seen in Fig. 22, the calculations agree reasonably well with the pressure history up to the point when the spall signal arrives at the gauge. This flyer plate experiment illustrates the contact surface algorithm can accurately model collision problems.

4.7. Exploding aluminum shell

A series of experiments were performed in [38] to determine the equation of state for the PBX9404 high explosive. These experiments will be used in this work to test the ability of the contact surface algorithm to capture collision and the subsequent push caused by the expanding gaseous products. The experiments have a ball of high explosive surrounded by a spherical aluminum shell. The experiments measured the outer radius of the aluminum shell as a function of time. The aluminum

![Fig. 23. The mesh is shown above for the exploding aluminum shell experiment with a 3.81 cm gap between the high explosive and the outer aluminum shell. The sub-captions show the corresponding time from detonation.](image)
shell has an inside radius of 15.240 cm and an outside radius of 15.875 cm. Each experiment in the test series used a different outer radii for the PBX9404 high explosive charge. The different radii created a void between the high explosive and the aluminum shell. The void thicknesses studied here are 0.0 cm, 2.0 cm, and 3.81 cm. The 0.0 cm gap corresponds to the high explosive charge being in direct contact with the aluminum shell at the start of the experiment. The largest void thickness in the test series was 3.81 cm. The radii of the aluminum shell are the same for all experiments in the test series, so the high explosive mass changes between experiments.

These experiments were modeled using two Lagrangian meshes separated by a contact surface. The mesh details are as follows. The high explosive used a 0.2 cm radial resolution and the aluminum shell used a 0.1 cm radial resolution. The angular resolution was $3.75^\circ$ except near the origin where the angular resolution was increased by a factor of 2. The mesh was coarsened near the origin to improve the aspect ratio of the zones. Coarse-graining the mesh creates zones with 5 nodes. The Riemann-like solver is used at every node in the mesh, including the nodes along the mesh coarsening boundaries. The mesh is shown in Fig. 23. The high explosive is detonated at the center (i.e. origin) of the charge. The burn front is propagated via the

![Fig. 24](image1.png)

**Fig. 24.** The plot above shows the outer radius of the aluminum shell as a function of time for the exploding aluminum shell experiment. The calculations are in reasonably good agreement with the experimental data.

![Fig. 25](image2.png)

**Fig. 25.** The bi-metallic shaped charge used in the comparison study is shown above. The bi-metallic shaped charge uses two metal liners. The inner liner is copper (blue) and the outer liner is aluminum (green). The high explosive (HE) charge is PBX9501 (red). The detonation point is denoted with a yellow circle. An aluminum plate is located on the bottom edge of the HE to prevent the gaseous products from flowing around the metal liners. (For interpretation of the references to color in this figure legend, the reader is referred to the web version of this article.)
The PBX9404 high explosive charge is modeled with a JWL equation of state [41,34], where the equation of state values were taken from [35]. The aluminum equation of state and strength model are identical to those used in the previous problems in this paper.

The Exploding aluminum shell results are compared with experimental data in Fig. 24. The contact surface exerts a force on the aluminum shell that causes the shell to accelerate in a manner consistent with the experiment. The match to the data is reasonably good and it is slightly better than the match published in [35]. This experiment further validates the accuracy of the new contact surface approach.

**Fig. 26.** The images above correspond to a time of 25 μs. The top and middle images provide a comparison of CCH (right) with SGH (left). The top image shows a comparison of the entire calculation and the middle image shows a comparison of the liners. The bottom image provides an enlarged view of the CCH calculation in the vicinity of the aluminum plate. The most obvious difference between the two calculations is the CCH scheme allowed more gaseous products to flow around the aluminum plate. In other words, the SGH calculation is stiff relative to the CCH calculation. Another difference is the CCH calculation produced more tangential flow in the aluminum liner. The bottom edge of the aluminum liner in the CCH calculation is approximately 1 mm below the aluminum liner in the SGH calculation. Lastly, the enlarged view of the CCH calculation (bottom image) illustrates the Riemann-like problem is stable on the 5-noded elements. The aluminum plate had 5-noded elements where one of the 5-noded elements was on the contact surface. As shown above, the 5-noded elements allowed the zone face with 3 nodes to bend. No mesh stability models were required with the CCH calculation.
Comparison studies were performed using CCH and SGH on two problems. The first comparison problem is a bi-metallic shaped charge [18]. The unique feature of a bi-metallic shaped charge is it has two metal liners. The details of the shaped charge calculated in this paper are shown in Fig. 25. The goal of modeling the bi-metallic shaped charge is to test the contact surface approach on a problem with significant deformation and interfacial slip between two metals. Currently, no experimental data is available for the shaped charge modeled here. However, future experiments are planned to acquire data for validating the CCH contact surface method. In lieu of data, the CCH results will be compared with SGH results.

The second comparison problem is an exploding cylinder into a plate, which is shown in Fig. 27. The goal of modeling an exploding cylinder into a plate is to test the ability of the contact surface approach to handle multidimensional impact, interfacial slip, and separation. No experimental data is available for the exploding cylinder into a plate calculation so CCH results are compared to SGH results. A brief overview of the SGH method is provided below.

The SGH approach used in the comparison studies is a compatible scheme developed by Burton [4] and also discussed in a later paper by Caramana et al. [5]. The SGH contact surface approach is a second-order accurate iterative method that is based on work by Zywicz and Puso [42]. The specific details of the SGH contact approach used here are provided by Kenamond and Bement in [21].

5.1. Bi-metallic shaped charge

The bi-metallic shaped charge has two free-standing spherically-shaped liners, copper and aluminum respectively, which are separated by a frictionless contact surface. The inner liner is made of copper and the outer liner is made of aluminum. The metal liners are driven by a PBX9501 conventional high explosive (HE) charge. An aluminum plate is located on the bottom edge of the shaped charge to prevent the gaseous products from flowing around the metal liners, which would otherwise tangle the Lagrangian mesh. The copper and aluminum use the same material models and corresponding values as the Taylor-Anvil experiments, the flyer plate experiments, the rod–rod impact problems, and the exploding aluminum shell experiments. The PBX9501 high explosive charge is modeled with a JWL equation of state [41,34], where the equation of state values were taken from Dobratz and Crawford [11]. The high explosive is detonated 9.5 cm from the origin of the liner and the burn front is propagated using the program burn model [27].

The mesh details are as follows. The copper linear had 6 radial zones, the aluminum liner had 12 radial zones, and the HE charge had 24 radial zones. The angular resolution was 90 zones, which created roughly square zones in metal liners at the beginning of the calculation. The zoning was coarsened by a factor of 2 in the radial direction in the aluminum plate, which created 5-noded elements. In the CCH calculation, the extra node will allow the zone to have curvature along the zone face.
that has 3 nodes. The motivation for including 5-noded elements is to test the nodal solver on a mix of element types and illustrate the robustness of the nodal solver.

In the case of CCH, a Riemann-like problem was solved at every node in the mesh including the nodes with reduced connectivity (i.e. three zones faces connected to the node versus four zone faces). The deformation of the 5-noded elements in the CCH calculation is shown in the bottom image of Fig. 26 corresponding to 25 μs after detonation. The Riemann-like solution is stable on the 5-noded elements in the interior mesh and the 5-noded element on the contact surface. In contrast to this CCH scheme, the stability on the 5-noded elements in SGH is achieved by constraining the degrees of freedom added by middle node on the zone face. Therefore, a 5-noded element in SGH will behave like a 4-noded element. In addition, mesh stability models, such as temporary quadrilateral subzoning [6], must be added to the SGH calculation to damp hourglass [14] and other null modes. Mesh stability models are not used in any CCH calculation in this work.

The deformation and flow of the bi-metallic shaped charge is shown in Fig. 26 corresponding to a time of 25 μs. The CCH results are on the right side and the SGH results are on the left side. As seen in Fig. 26, the contact surface approach allowed the aluminum liner to flow tangentially around the copper liner. The deformation and the shear flow in the metal liners are similar in the two calculations; however, small differences are noticeable. The observable differences between the approaches are (1) the aluminum liner slide more tangentially along the contact surface in the CCH calculation than the SGH approach, and (2) the CCH calculation allowed more gaseous products to flow around the aluminum plate than SGH. The results from this comparison study illustrate the new CCH contact surface approach is a viable way to calculate complex hydrodynamic experiments.
5.2. Exploding cylinder into a plate

The exploding cylinder into a plate problem has a copper box surrounding a PBX9501 conventional HE charge. The copper box expands and collides with an aluminum plate. The copper cylinder and aluminum plate are separated by a frictionless contact surface. The copper and aluminum use the same material models and corresponding values as the Taylor-Anvil experiments, the flyer plate experiments, the rod–rod impact problems, the exploding aluminum shell experiments, and the bi–metallic shaped charge problem. The high explosive is detonated at the origin and the burn front is propagated using the program burn model. The initial configuration is discretized using a uniform 0.05 cm mesh resolution.

The deformation and material flow in the exploding cylinder into a plate problem is shown in Fig. 28. The CCH results are on the right side and the SGH results are on the left side. As seen in Fig. 28, the contact surface approach allows the copper cylinder to impact and slide along the aluminum plate; furthermore, the aluminum plate is able to separate from the copper cylinder after the collision. The deformation, the shear flow, and the separation between the metals are similar in the two calculations; however, small differences are noticeable. For example, the SGH calculation produced a slightly larger separation gap between the copper cylinder and the aluminum plate at the axis of rotation. The separation difference might be the result of treating the contact surface algorithm as first-order accurate. Despite small differences, the CCH and SGH calculations show similar behavior. The results from this comparison problem further illustrate the viability of the new CCH contact surface approach.

6. Conclusion

A new frictionless contact surface approach for CCH was presented that allows surfaces to impact, slide, and separate. The velocity of a penetrating node on the contact surface and the corresponding forces are calculated using a multi-directional Riemann-like nodal solver based on the work of Burton et al. [3]. The Riemann-like problem includes information from both surfaces. The corner impedance, corner velocity, and corner stress of the contacting node set are mapped to the penetrating node. Surface area weights are used to account for the difference in the iota surface area between the contacting nodes and the penetrating node. The approach is applied to every penetrating node along the contact surface.

The new contact surface approach was first verified using test problems with analytic solutions. The test problems were a one-dimensional slab impact problem, planar Cartesian Noh, two-dimensional XY Noh, and two-dimensional XY Sedov. These test problems were performed with contact surfaces. In additional calculations were performed with aligned and unaligned meshes along the contact surface. The calculations of these test problems were in good agreement with the analytic solutions. Next, a series of experiments were calculated to verify the contact surface algorithm on more complex problems. The first experiments calculated were two Taylor-Anvil experiments. The Taylor-Anvil experiments used to test the ability of the algorithm to handle both collision and the subsequent tangential slip along the contact surface. Next, two rod–rod impact problems were calculated to demonstrate the symmetry of the contact surface approach. The rod–rod calculations illustrate the ability of two Lagrangian meshes to separate. The next experiment calculated was a flyer plate experiment. The calculated pressure history was compared to the experimental data. The last set of experiments were three exploding aluminum shells. These experiments were used to validate the ability of the contact surface approach to model impact and the subsequent push caused by the expanding gases. Lastly, comparison studies were performed using a bi–metallic shaped charge and an exploding cylinder into a plate. The goal of the comparison studies is to compare the CCH results with SGH results on problems with complex flows along the contact surface combined with significant deformation in the materials. The calculated deformation in the metallic materials using CCH was in good agreement with SGH results. Most importantly, the comparison studies illustrate CCH is capable of modeling complex hydrodynamic experiments including problems where contact surfaces are present.

Acknowledgments

We gratefully acknowledge the support of the National Nuclear Security Administration through the Laboratory Directed Research and Development (LDRD) program and the Advanced Simulation and Computing (ASC) program. We also thank Misha Shashkov for his insightful suggestions. The Los Alamos National Laboratory unlimited release number is: LA-UR-13-21631.

References


