

Experimental Validation of a Coupled Fluid-Multibody Dynamics Model for Tanker Trucks

Tamer M. Wasfy

Advanced Science and Automation Corp.

James O'Kins and Scott Smith

U.S. Army Tank Automotive Research Development and Engineering Center

ABSTRACT

A time-accurate finite element model for predicting the coupled dynamic response of tanker trucks and their liquid payloads is presented along with an experimental validation of the model. The tanker truck components are modeled using rigid bodies, flexible bodies, joints and actuators. The model is validated using a full-scale army heavy class tactical trailer carrying a water tank. The trailer is placed on an n-post motion base simulator which was used to perform harmonic/ramp pitch, roll and stir excitations of the trailer in order to simulate typical road maneuvers. Experiments were carried out with an empty tank and a 65%-filled tank. The time-histories of the tires and suspension system deflections are measured for the various input motion excitations. The experiment's measurements are compared with the results predicted using the computational model. The comparison shows that the model can predict with reasonably good accuracy the test tanker-trailer's dynamic response.

1. INTRODUCTION

A time-accurate finite element model for predicting the coupled dynamic response of tanker trucks and their liquid payloads is presented along with an experimental validation of the model. Tanker trucks are used to transport liquids (such as fuels and water) from production facilities to distribution outlets. They provide the most flexible and readily deployable form of liquid transportation. Tanker trucks usually consist of a truck that may be pulling one or more trailers with one or more tanks carried on the truck and/or the trailer(s). Figure 1 shows a typical military tanker truck system consisting of a truck pulling a trailer that is carrying a potable water tank.

Many tanker truck accidents occur due to sloshing of the liquid payload when the tank is nearly half-full. The sloshing can shift the center of mass of the tank and can cause high time-varying inertia forces. This can cause the tanker truck to overturn, lose traction on one or more

tires, or be hard to brake or steer. All this can result in tanker-truck accidents which are costly and dangerous especially if the tank contains hazardous liquids (such as fuels, petrochemicals, or toxic liquids.) The estimated yearly cost of truck accidents in the US is about \$1.2 billion for trucks carrying hazardous material and \$43 billion for trucks carrying non-hazardous material [1]. The majority of those accidents (more than 75%) occur en-route. Accidents involving fire trucks account for approximately 20% of U.S. fire fighter deaths each year. Crashes involving fire trucks are the most prevalent of these types of accidents with 87% occurring when the fire truck went out of control and rolled over or left the road (no collision) [2]. The cost of physically building and testing tanker trucks can be very high. Several configurations with various shapes and locations for the baffles within the tanks, different tank geometries, and suspension system characteristics need to be tested. Considerable savings and better performance can be achieved if most of the testing is done numerically. The model presented and validated in this paper can help increase the stability, safety margins, maximum allowable speed, and/or liquid carrying capacity of tanker trucks and can help to reduce the annual costs of tanker truck accidents. It can also be used to investigate the cause of tanker truck accidents and to come up with the most effective design improvements that can increase the safety of tanker trucks.

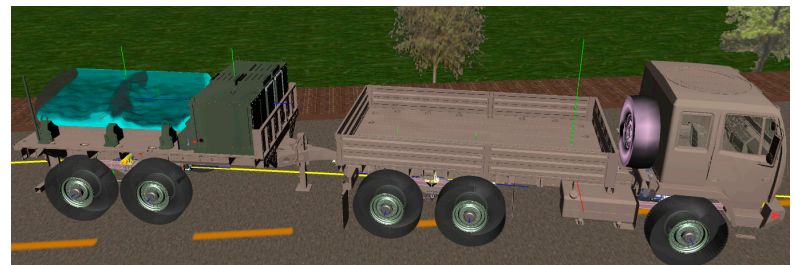


Figure 1 An FMTV (Family of Medium Tactical Vehicles) military tanker truck system consisting of a truck pulling a trailer that is carrying a water tank.

The model can also be easily extended to other applications involving flexible multibody systems carrying liquid filled tanks such as: tankers, tankers, tanker ships, passenger cars (fuel-tanks), airplanes

(fuel-tanks and payload tanks), space launch vehicles (liquid fuel tanks), and satellites/space structures (liquid fuel tanks).

The experimental validation of the numerical model was carried out using a full-scale PLS (Palletized Load System) tanker-trailer that was instrumented and mounted on an n-post motion simulator (Figure 2). The n-post motion simulator consists of hydraulic linear actuators under each tire that can be independently controlled to simulate typical road maneuvers that the vehicle will be subjected to such as lane-change and going over symmetric or asymmetric bumps/potholes. Experiments were carried out with an empty tank and a 65% filled tank. The time-histories of the tires and suspension system deflections are measured for various input motion excitations. The measurements are compared with the results predicted using the computational model.



Figure 2 Instrumented PLS (Palletized Load System) tanker-trailer mounted on an n-post motion simulator that was used to validate the present finite element model.

In order to accurately predict the dynamic response of tanker trucks, the model must accurately account for the following effects:

- Incompressible liquid flow in a moving/deforming container (governed by the incompressible Navier-Stokes equations).
- Modeling of the liquid's free-surface.
- Modeling of fluid viscosity and turbulence.
- Coupling between the solid and the fluid at the fluid-structure interface.
- Large rotation of the solid bodies.
- Deformation of the solid flexible bodies. Flexible bodies can be modeled as springs, beam, shells or general solids. For example a general solid model can be used for leaf-springs while a linear-spring

model can be used to model struts and suspension springs.

- Joint kinematic constraints for spherical, revolute, cylindrical and prismatic joints including joint friction and clearances.
- Tire load-deflection and damping characteristics.
- Frictional contact between the tire and the terrain.
- Actuators and control laws.
- Motion control components including transmission components, clutches, and brakes (All those components involve friction).

In the present model, the fluid governing equations of motion are the incompressible Arbitrary Lagrangian-Eulerian Navier-Stokes equations along with a large-eddy simulation (LES) turbulence model. Incompressible fluid flow can be modeled using finite volume, finite element or particle methods. In addition, a technique for modeling the free-liquid surface must also be used. A literature review of the techniques for modeling fluid flow with a free-surface was presented in [3]. Those techniques are:

- Volume-of-fluid (VOF) method* [4-9]. Each element has a VOF value between 0 (for empty elements) and 1 (for elements completely filled with fluid). The free surface is reconstructed for each element using piecewise-linear planar segments that are calculated from the VOF value of the element along with the VOF values of neighboring elements (which are used to determine the normal to the planar surface).
- Level-set method*. This method uses a smooth scalar function defined at every node in the fluid domain which specifies the signed smallest Euclidean distance between the node and the interface [10]. The evolution of the scalar function is governed by a convection transport equation where the interface is moved with the fluid velocity.
- Arbitrary Lagrangian-Eulerian (ALE) method*. Using this method, the fluid mesh deforms and moves along with the fluid's free-surface [11-16]. A disadvantage of this method is that it does not allow large surface deformation including surface break-up and merging unless the fluid domain is re-meshed [14, 15].
- Lagrangian particle methods*. Lagrangian particles which represent small packets of fluid are used to model the fluid flow. A contact model between the fluid particles is used to model the fluid compressibility and viscous effects. A special type of this class of methods which has been successfully applied to free-surface flows and fluid-structure interaction problems is the particle finite element method (PFEM) [17] in which the particles are used to generate a polyhedral finite element mesh every time step using an extended Delaunay tessellation. The solution of the incompressible Navier-Stokes equations is then carried out using that mesh.

Techniques to handle fluid flow in a moving/deforming container were also reviewed in [3]. They include:

1. *Fixed Cartesian fluid mesh with cut-cell boundary condition.* The fluid domain is a Cartesian mesh [4, 5]. The container moves inside this mesh. A cut-cell technique is used to find where the boundary of the container intersects with the fluid cells.
2. *Moving ALE mesh.* The fluid is modeled using a fluid mesh that moves and deforms with the tank [3, 9, 11-13].
3. *Fixed-fluid mesh with the Navier-Stokes equations written in a reference frame fixed to the tank* [11, 18].
4. *Particle methods* [17]. Normal contact constraint between the particles and the tank wall is used to model the wall impenetrability constraint. Tangential friction between the particles and the tank wall is used to model wall adhesion and viscous effects.

In order to model fluid flow with a free-surface in a moving deforming container, the above methods must be combined. Table 1 shows the references where the various combinations of the above techniques were used.

Table 1 References for the various combinations of techniques for modeling a free-surface flow in a moving deforming container.

Free surface model	Moving / deforming tank model	Fixed grid	ALE	NS written in tank frame	Particles
VOF		4, 5	3, 9 (2D) Present paper	18	
Level-set			10		
ALE			11, 12, 13, 14, 15, 16	11	
Particles					17

In the present paper, an acceptor-donor VOF based algorithm is used to model the free-surface and the ALE method is used for modeling fluid flow in a moving/deforming container. Using the VOF technique for modeling the free-surface has the following advantages over other methods:

- The VOF technique can conserve mass and energy better than the level-set method. This is due to the fact that the VOF technique explicitly balances local mass transfers between elements. On the other hand, the level-set method relies on the fluid flow solution to conserve mass and uses interpolated velocities which can lead to accumulation of mass loss or gain errors.
- Unlike the ALE technique, the VOF technique can handle fluid breakup and merging without the need for re-meshing. If re-meshing is used frequently, it can degrade the solution accuracy due to re-interpolation of the solution field onto the new mesh.
- Particle methods require a large number of particles to accurately model the free surface. The PFEM requires less fluid particles, however, the tessellation step is computationally intensive.

Using the ALE method for modeling the moving/deforming container has the following advantages over other methods:

- In the fixed Cartesian fluid mesh method, since the solid container cuts the mesh cells at arbitrary surfaces, the fluid-solid impenetrability and no-slip boundary conditions are satisfied only in a time average sense. Also, the method has stability and accuracy problems when the cut-cell elements at the solid-fluid interface become small. In addition, for tanker truck applications, since the tank can undergo very large translation as the truck is moving on the road, the Cartesian mesh must be moved with the tank. This means the fluid mesh is no longer fixed, and that an ALE formulation must be used to move the mesh. This reduces the simplicity of the method.
- The main disadvantage of using a fixed-fluid mesh with the Navier-Stokes equations written in a reference frame fixed to the tank is that since the tank frame is a non-inertial frame (accelerating frame), writing the equations of motion with respect to that frame results in a complex inertia operator which involves centrifugal and Coriolis acceleration terms. Also, this method cannot - by itself - deal with a deforming container.

A review of multibody dynamics modeling techniques including deformation reference frames, treatment of large rotations, discretization techniques, finite elements, constraint and contact modeling, and solution techniques is presented in [19]. In the present paper, a flexible multibody dynamics code with the following characteristics is used:

- The solid, fluid and fluid-solid interface governing equations of motion are solved along with joint/constraint equations using a time-accurate explicit solution procedure that can maintain time-accuracy over long simulation times. The explicit solution procedure was presented in Ref. [3] and, for the sake of completeness, is outlined in the Appendix.
- Total Lagrangian, total displacement equations of motion formulation with the degrees of freedom referred to a global inertial reference frame [15-18].
- Flexible bodies can be modeled using a library of spring, truss, beam, and solid nonlinear finite elements with Cartesian coordinate degrees of freedom. This allows arbitrarily large element rotations. The elements library includes:
 - Struts and suspension springs, that are modeled using a linear spring element, that can include a prescribed stiffness-deflection relation, a damping-deflection rate relation and a friction coefficient-deflection rate relation.
 - Torsional-spring type 3-node beam elements [20, 21].
 - Natural-modes eight-node brick elements [22, 23]. Those elements can also be used

to model shells and beams. One element through the thickness is sufficient to accurately model the membrane, shear, and bending characteristics. They do not exhibit locking or spurious modes (widely used techniques to alleviate locking such as hourglass control lead to elements that do not maintain solution accuracy over very long solution times). They are computationally efficient. Assumed strain elements of comparable accuracy are more computationally expensive. Any material law can be used with those elements including: linear elastic, hyper-elastic, and non-linear laws. Leaf-springs are modeled using the natural-modes brick elements.

- Connection points on the rigid and flexible bodies are used to define joints between the bodies including spherical, revolute, cylindrical, prismatic and rigid joints. A penalty model is used to impose the joint/contact constraints [24]. An asperity-based friction model is used to model joint/contact friction [25].
- Rigid body rotational equations of motion are written in a body (material) frame, with the resulting incremental rotations added to the total body rotation matrix [24]. The rigid bodies rotational equations of motion are written in a body-fixed frame with the total rigid-body rotation matrix updated each time step using incremental rotations.
- Normal contact is modeled using a penalty formulation [26, 27]. Frictional contact is modeled using an accurate and efficient asperity-based friction model [25]. Tires are modeled using a one node tire model that accounts for distributed tire normal contact using the penalty formulation and friction using the asperity-based friction model.
- Special elements are used to model wheels/pulleys [26, 27], sprockets [28], and clutches [29].
- The code uses a general contact search algorithm that finds the contact penetration between finite elements and other elements as well as general triangle and quadrilateral surfaces.

Two-way coupling between the multibody system (vehicle) motion and the fluid is achieved by satisfying the following conditions at the solid-fluid interface:

- The fluid velocity normal to the solid's surface must be equal to the normal solid velocity.
- The fluid velocity tangent to the solid surface can range from being equal to the tangential velocity of the solid surface (no slip condition) to being free.
- No additional energy or momentum to the system should be introduced at the interface.

The above boundary conditions are satisfied by solving Newton's equations of motion at the fluid-solid interface nodes for a common normal acceleration for the fluid

and the solid at the interface. The tangential fluid and solid accelerations can range from being the same (no-slip condition) to being completely decoupled.

In the present paper, a single computational code which uses a time-accurate explicit solution procedure is used to solve both the solid and fluid equations of motion. Many commercial software and studies on modeling liquid sloshing coupled with solid body motion use two codes which pass the interface forces and motion back and forth and iterate on the two codes until equilibrium is achieved [e.g. 6, 18]. This approach adds extra computational burden and, in general, does not achieve the same accuracy as the single integrated code solution due to the difficulty in achieving an equilibrium solution between two disjointed codes.

The rest of this paper is organized as follows. In Section 2 the equations of motion for the solid, fluid and fluid-structure interface are presented. In Section 3 the frictional contact model is presented along with a description of the 1-node distributed contact tire model. In Section 4 the VOF free-surface model is presented. In Section 5 the experimental setup along with the validation study are presented. Finally, in Section 6 concluding remarks are offered.

2. EQUATIONS OF MOTION

In the subsequent equations the following conventions will be used:

- The indicial notation is used.
- The Einstein summation convention is used for repeated subscript indices unless otherwise noted.
- Upper case subscript indices denote node numbers.
- Lower case subscript indices denote vector component number.
- The superscript denotes time.
- A superposed dot denotes a time derivative.

2.1 SOLID EQUATIONS OF MOTION

The solid translational equations of motion are written with respect to the global inertial reference frame and are obtained by assembling the element equations. The finite elements used here use only translational DOFs with no rotational DOFs. The translational equations of motion include the rigid-body translational DOFs. They can be written as:

$$M_K \ddot{x}_{Ki}^t = F_{sKi}^t + F_{aKi}^t \quad (1)$$

where t is the running time, K is the global node number (no summation over K ; $K=1 \rightarrow N$ where N is the total number of nodes), i is the coordinate number ($i=1,2,3$), a superposed dot indicates a time derivative, M_K is the lumped mass of node K , x is the vector of nodal Cartesian coordinates with respect to the global inertial reference frame, and $\ddot{x}$ is the vector of nodal accelerations with respect to the global inertial reference frame, F_s is the vector of internal structural forces, and F_a is the vector of externally applied forces, which include surface forces and body forces.

For each rigid body, a body-fixed material frame is defined. The rigid body is represented by one node located at the body's center of mass, which is also the origin of this frame. The mass of the body is concentrated at the node and the inertia of the body, given by the inertia tensor I_{ij} , is defined with respect to the body frame. The orientation of the body-frame is given by $R_K^{t_0}$ which is the rotation matrix relative to the global inertial frame at time t_0 . The rotational equations of motions are written for each rigid body with respect to its body-fixed material frames as:

$$I_{Kij} \ddot{\theta}_{Kj}^t = T_{sKi}^t + T_{aKi}^t - (\dot{\theta}_{Ki}^t \times (I_{Kij} \dot{\theta}_{Kj}^t))_{Ki} \quad (2)$$

where I_K is the inertia tensor of rigid body K , $\ddot{\theta}_{Kj}$ and $\dot{\theta}_{Kj}$ are the angular acceleration and velocity vectors' components for rigid body K relative to its material frame in direction j , T_{sKi}^t is the component of the vector of internal torque at node K in direction i , and T_{aKi}^t is the component of the vector of applied torque in direction i . The summation convention is used only for the lower case indices i and j .

The trapezoidal rule is used as the time integration formula for solving equation (1) for the global nodal positions x :

$$\dot{x}_{Kj}^t = \dot{x}_{Kj}^{t-\Delta t} + 0.5 \Delta t (\ddot{x}_{Kj}^t + \ddot{x}_{Kj}^{t-\Delta t}) \quad (3a)$$

$$x_{Kj}^t = x_{Kj}^{t-\Delta t} + 0.5 \Delta t (\dot{x}_{Kj}^t + \dot{x}_{Kj}^{t-\Delta t}) \quad (3b)$$

where Δt is the time step. The trapezoidal rule is also used as the time integration formula for the nodal rotation increments:

$$\dot{\theta}_{Kj}^t = \dot{\theta}_{Kj}^{t-\Delta t} + 0.5 \Delta t (\ddot{\theta}_{Kj}^t + \ddot{\theta}_{Kj}^{t-\Delta t}) \quad (4a)$$

$$\Delta \theta_{Kj}^t = 0.5 \Delta t (\dot{\theta}_{Kj}^t + \dot{\theta}_{Kj}^{t-\Delta t}) \quad (4b)$$

where $\Delta \theta_{Kj}$ are the incremental rotation angles around the three body axes for body K . The rotation matrix of body K (R_K) is then evaluated using:

$$R_K^t = R_K^{t-\Delta t} R(\Delta \theta_{Kj}^t) \quad (5)$$

where $R(\Delta \theta_{Kj}^t)$ is the rotation matrix corresponding to the incremental rotation angles from Equation (4b).

The explicit solution procedure used for solving equations (1-5) along with the constraint equations is presented in Section 6. The constraint equations are generally algebraic equations, which describe the position or velocity of some of the nodes. They include:

- Prescribed motion constraints:

$$f(\{x\}, t) = 0 \quad (6)$$

- Joint constraints:

$$f(\{x\}) = 0 \quad (7)$$

- Contact/impact constraints:

$$f(\{x\}) \geq 0 \quad (8)$$

The penalty technique is used for imposing the constraints in which a normal reaction force is generated when a node penetrates into a contact body. The magnitude of the force is proportional to the penetration distance [24-26].

2.2 FLUID EQUATIONS OF MOTION

The dynamic response of the fluid is described by the ALE version of the incompressible Navier-Stokes equations. Namely, the equations of conservation of momentum and mass for a moving deforming control volume along with a large-eddy simulation (LES) transport equation. Those are:

$$\int_V \rho \frac{\partial \hat{u}_i}{\partial t} dV = \int_V -u_i \frac{\partial \rho}{\partial t} dV + \int_V \frac{\partial [-\rho u_i \hat{u}_j - P \delta_{ij} + \tau_{ij}]}{\partial x_j} dV + \int_V \rho f_i dV \quad (9)$$

$$\int_V \frac{\partial \rho}{\partial t} dV + \int_V \frac{\partial (\rho \hat{u}_i)}{\partial x_i} dV = 0 \quad (10)$$

$$\rho = \rho_0 + r P \quad (11)$$

$$\hat{u}_i = u_i - v_i \quad (12)$$

$$\tau_{ij} = \lambda D_{kk} \delta_{ij} + 2(\mu + \mu_t) D_{ij} \quad (13a)$$

$$D_{ij} = 0.5(\partial \hat{u}_i / \partial x_j + \partial \hat{u}_j / \partial x_i) \quad (13b)$$

$$\int_V \frac{\partial (\rho K)}{\partial t} dV = - \int_V \frac{\partial (\rho \hat{u}_i K)}{\partial x_i} dV + \int_V \frac{\partial}{\partial x_i} \left[(\mu + \mu_t) \frac{\partial K}{\partial x_i} \right] dV + \quad (14)$$

$$\int_V \left(2\mu_t D_{ij} - \frac{2}{3} \rho K \partial_{ij} \right) \frac{\partial \hat{u}_i}{\partial x_j} dV - \int_V \rho \varepsilon dV$$

$$\varepsilon = 1.0 K^{3/2} \quad (15a)$$

$$\mu_t = 0.05 \sqrt{K} \quad (15b)$$

where V is the element volume, t is the running time, ρ is the density of the fluid, $\vec{u}$ is the fluid velocity vector relative to the global reference frame, $\vec{\hat{u}}$ is the fluid velocity vector relative to the moving fluid mesh, $\vec{v}$ is the velocity of the fluid mesh, P is the relative pressure, $\vec{x}$ is the position vector, τ is the deviatoric stress tensor, D is the rate of deformation tensor, $\vec{f}$ is the body force vector, r is the artificial compressibility parameter, ρ_0 is the nominal fluid density, μ is the fluid viscosity, μ_t is an additional turbulence viscosity calculated using Equation 15b), K is the eddy kinetic energy, and ε is the sub-grid scale eddy kinetic energy dissipation term. Incompressible flow is modeled using the artificial compressibility technique [30]. A finite element formulation is used to derive the element's semi-discrete equations of motion from the governing equations (9-13). 8-node hexahedral elements are used with tri-linear equal-order velocity and pressure interpolation. A pressure averaging algorithm [31] is used to eliminate pressure checker-boarding (due to the use of an equal order interpolation for pressure and velocity). The element equations are assembled into the global semi-discrete equations of motion:

$$M_{fN} \dot{u}_{Ni}^t = F_{fNi}^t \quad (16)$$

$$V_{fN} \dot{P}_N^t = Q_{fN}^t \quad (17)$$

$$M_{fK} \dot{K}_N^t = S_{fN}^t \quad (18)$$

where M_{fN} is the lumped fluid mass of node N , $\dot{u}_{Ni}^t$ is component i of the fluid acceleration at node N , F_{fNi}^t is component i of the fluid forces at node N , V_{fN} is the lumped fluid volume at node N , $\dot{P}_N^t$ is the fluid pressure rate at node N , Q_{fN}^t is the fluid pressure flux at node N , $\dot{K}_N^t$ is the eddy kinetic energy rate at node N , and S_{fN}^t is the eddy kinetic energy flux at node N . Those equations are integrated using the trapezoidal rule along with an

explicit solution procedure to yield the nodal fluid velocity and pressure:

$$u_{Kj}^t = u_{Kj}^{t-\Delta t} + 0.5 \Delta t (\dot{u}_{Kj}^t + \dot{u}_{Kj}^{t-\Delta t}) \quad (19)$$

$$P_{Kj}^t = P_{Kj}^{t-\Delta t} + 0.5 \Delta t (\dot{P}_{Kj}^t + \dot{P}_{Kj}^{t-\Delta t}) \quad (20)$$

$$K_{Kj}^t = K_{Kj}^{t-\Delta t} + 0.5 \Delta t (\dot{K}_{Kj}^t + \dot{K}_{Kj}^{t-\Delta t}) \quad (21)$$

2.3 FLUID-STRUCTURE INTERFACE EQUATIONS OF MOTION

Newton's equations of motion are used to find a common normal acceleration for the fluid and the solid at the interface. This is done for each node at the fluid-structure boundary as follows:

$$(m_s + m_f) \ddot{u}_n = \sum \text{FluidForces} + \sum \text{StructureForces} \quad (22a)$$

$$(m_s + m_f) \ddot{v}_n = \sum \text{FluidForces} + \sum \text{StructureForces} \quad (22b)$$

where m_s is the solid mass of the node, m_f is the fluid mass of the node, $\ddot{u}_n$ and $\ddot{v}_n$ are respectively the fluid and solid accelerations of the node normal to the fluid-structure interface. The tangential fluid and solid accelerations ($\ddot{u}_t, \ddot{v}_t$) are calculated using the following equations:

$$((1-s)m_s + m_f) \ddot{u}_t = (1-s) \sum \text{StructureForces} + \sum \text{FluidForces} \quad (23a)$$

$$(m_s + (1-s)m_f) \ddot{v}_t = \sum \text{StructureForces} + (1-s) \sum \text{FluidForces} \quad (23b)$$

where s is the slip factor. A no-slip condition corresponds to a slip factor of zero. The slip factor determines how much of the fluid and structure forces are mutually exchanged. Equations 22 and 23 are written for all fluid-solid interface nodes. The fluid mesh must move and deform with the tank. This is done by modeling the fluid mesh using very light and compliant (3 orders of magnitude less than the tank) solid brick elements (called "mock" mesh). The ALE formulation is used to account for the fluid mesh deformation/motion.

3. TIRE FRICTIONAL CONTACT MODEL

A tire is mounted on a rigid body representing the wheel. Contact is detected between the tire's surface and a polygonal surface representing the contact body. The tire's external surface is discretized in the circumference and meridian directions into a grid of rectangles with a contact point defined at the center of each rectangle (Figure 3). The global position (x_{Gcp}) and velocity ($\dot{x}_{Gcp}$) of a contact point is obtained using the rigid body position and rotation matrix:

$$x_{Gcp_i} = X_{BF_i} + R_{BF_{ij}} x_{Lcp_j} \quad (24)$$

$$\dot{x}_{Gcp_i} = \dot{X}_{BF_i} + R_{BF_{ij}} (W_{BF} \times x_{Lcp})_j \quad (25)$$

where X_{BF} and $\dot{X}_{BF}$ are the global position and velocity vectors of the wheel's frame, R_{BF} is the rotation matrix of the wheel relative to the global reference frame, W_{BF} is the wheel's angular velocity vector relative the local wheel's frame, and x_{Lcp} is the position of the contact point relative to the wheel's frame. The frictional contact force at each contact point (sum of the normal contact and tangential friction forces) is transferred as a force and a moment to the center of the wheel. The negative of this force is transferred to the center of the contact

body. Those forces and moments are summed and applied as external forces to the wheel and contact body.

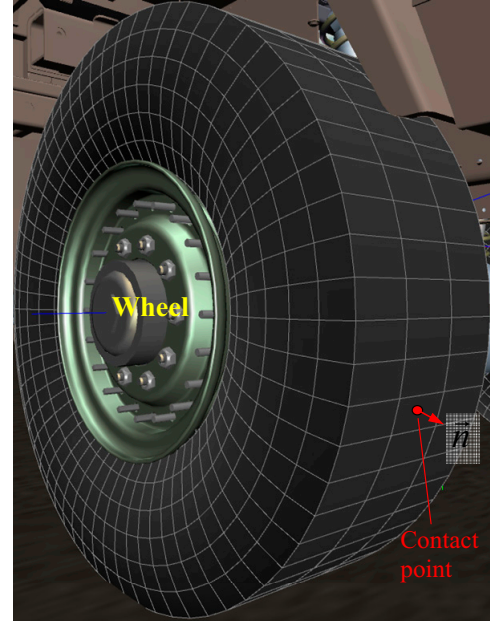


Figure 3 Tire mounted on a wheel and discretized into a grid of rectangular elements with a typical contact point at the center of an element.

3.1 PENALTY NORMAL CONTACT MODEL

The penalty technique is used for imposing the contact constraints. In this technique, a normal reaction force (F_{normal}) is generated when a contact point is in contact with a body (i.e. when the point is inside the body) that is given by [25, 26]:

$$F_{normal} = A k_p d + A \begin{cases} c_p \dot{d} & \dot{d} \geq 0 \\ s_p c_p \dot{d} & \dot{d} < 0 \end{cases} \quad (26)$$

where A is the area of the rectangle associated with the contact point, k_p and c_p are the penalty stiffness and damping coefficient per unit area; d is the closest distance between the node and the contact surface (Figure 4); s_p is a separation damping factor between 0 and 1 which determines the amount of sticking between the contact node and the contact surface at the node (leaving the body). The normal contact force vector is given by:

$$F_{n_i} = n_i F_{normal} \quad (27)$$

where $\vec{n}$ is the normal to the surface. The total force on the node generated due to the frictional contact between the point and surface is given by:

$$F_{po \text{ int } i} = F_{t_i} + F_{n_i} \quad (28)$$

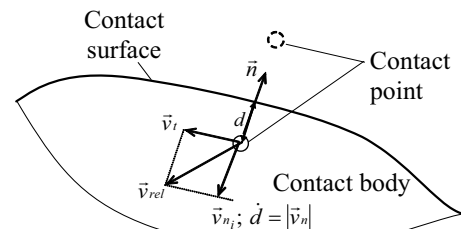


Figure 4 Contact surface and contact node.