Three-dimensional, Multi-Phase CFD Modeling using an Overset Mesh with Free Surface to Simulate Wave Impacts on the Outer Shell of a Spacecraft

THERMAL & FLUIDS
ANALYSIS WORKSHOP
Ames Research Center 2025

Presented by: Bill Dziedzic
NASA Kennedy Space Center, FL

Thermal & Fluids Analysis Workshop 2025
NASA Ames Research Center

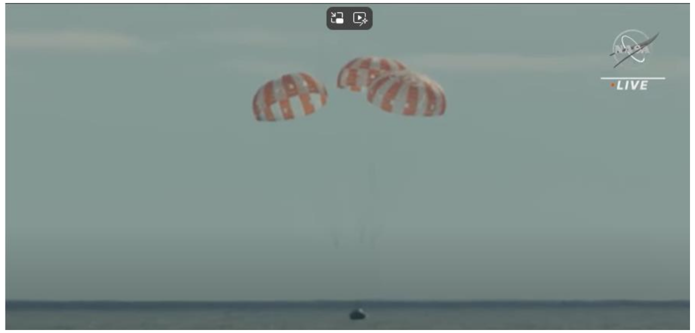
San Jose, CA
August 4-7, 2025

Background

Artemis I Orion spacecraft splashdown on Dec 11, 2022.

After 25.5 days in space, and a 1.3-million-mile (2.1-million-km) journey around the Moon, Orion splash down in the Pacific Ocean off the coast of California at 12:39 p.m. EST (17:39 UTC) on Sunday, Dec. 11.

The exploration ground systems recovery team from NASA's Kennedy Space Center in Florida, working with the U.S. Navy, recovered the spacecraft.



Splashdown approx.: 1:40:41

https://youtu.be/xzZPzmMtQA8

Tow time approx. 0.33

https://youtu.be/4FbuN-QugM

Traveling through space at about 25,000 miles per hour, the spacecraft will slow to 300 mph after it passes through the Earth's atmosphere.

The spacecraft then slows down to 20 mph before it safely splashes down in the Pacific Ocean.

Introduction

A Crew Module Test Article is used to simulate the Orion Multi-Purpose Crew Vehicle (Orion MPCV) in recovery training missions.

The Crew Module Test Article (CMTA), the CMTA is a full-scale mockup of the Orion spacecraft, is seen in the Pacific Ocean as teams practice Artemis recovery operations during Underway Recovery Test-12 onboard USS Somerset off the coast of California, Saturday, March 29, 2025. NASA/Bill Ingalls[6].

Tow time approx. time 0.33
https://youtu.be/_4FbuN-QugM

The objectives of this analysis was to simulate the wave impacts (pressure) on a spacecraft's outer shell under simulated sea conditions.

- This analysis will help provide a record of seakeeping load cases so that follow-on structural analysis can use the pressure results to help determine the stress and distortion on the spacecraft training article hull.

Abstract

This analysis describes the use of a three-dimensional, multi-phase, free surface transient computational fluid dynamics (CFD) model (Star CCM+) to simulate the wave impacts (pressure) on NASA's Artemis Crew Module Test Article (CMTA) outer shell under simulated sea conditions.

- The CFD analysis results helped determine pressure loads on the hull of a spacecraft test article that would assess the seakeeping/recovery effort for the Artemis lunar exploration program.
- CFD simulations of different wave heights, spacecraft under tow, and a simulated parachute water drop analysis are presented.
- The CFD study simulates the behavior of two fluids (air and water) within the same continuum incorporating the Eulerian multiphase model.
- The Volume of Fluid (VOF) Eulerian multiphase model is used to predict the distribution and movement of the water/air interface on the spacecraft.
- The Star CCM+ Overset Mesh feature and Dynamic Fluid-Body Interaction (DFBI) models are used to simulate the motion of the spacecraft on the water surface under different types of wave interactions and water impact from a simulated drop test.

CFD Modeling

The spacecraft is treated as a rigid body in response to applied forces and moments.

- The motion of this rigid body is free to rotate and translate in all directions allowing six degrees of freedom.
  - In a rigid body, the relative distance between internal points do not change.
- Therefore, the equations of motion for the center of mass of the body can be used.
- A continuum treated as a rigid body is coupled with a fluid boundary.
- The solver calculates the motion of the body in response to the fluid forces and moments at the coupled boundary.

This study involves three dimensional, implicit unsteady, turbulent Flow.

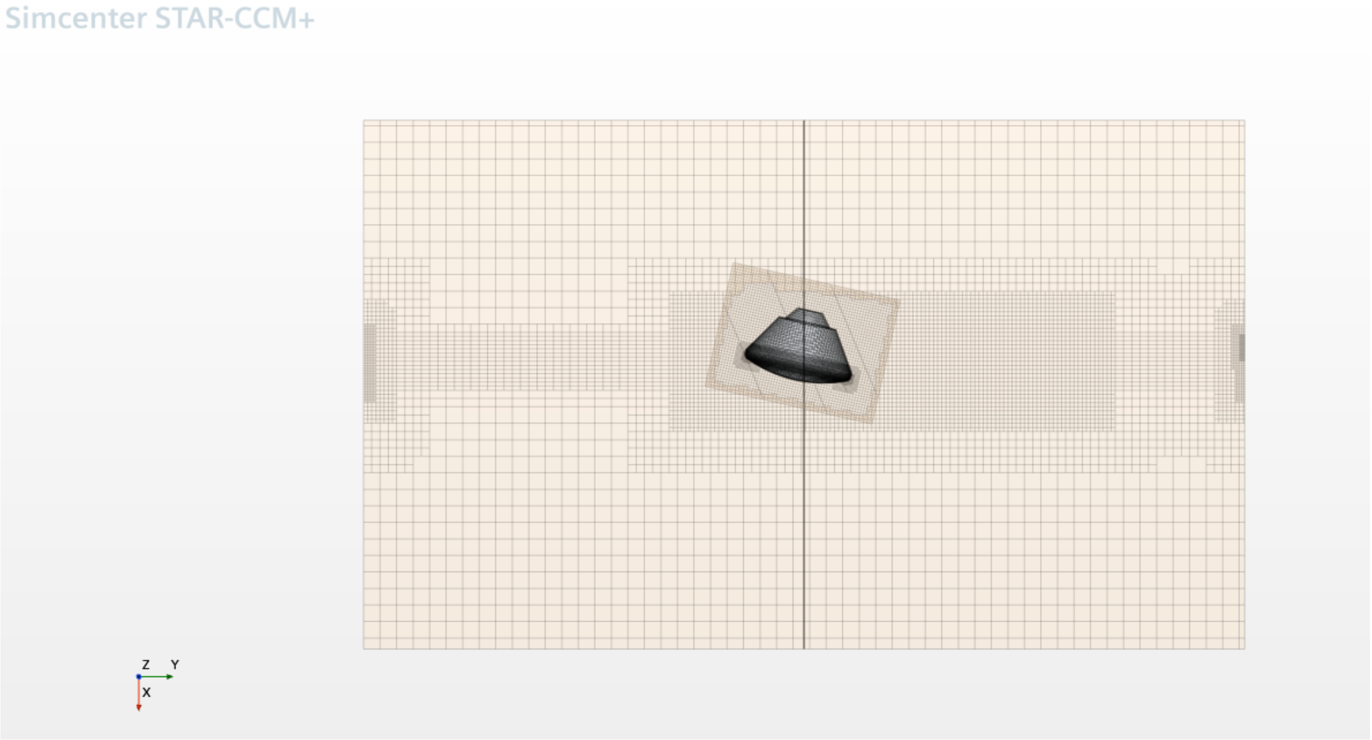
- The Reynolds-Averaged Navier-Stokes, Realizable K-Epsilon Turbulence, Two-Layer All Y+ Wall Treatment was used.
- Also other solver options used included Volume of Fluid, Eulerian Multiphase, Multiphase Equation of State, Multiphase interaction, Gravity, Gradients, and Segregated Flow.

The multiphase segregated flow model treats each phase as inter-penetrating continua.

- It solves transport equations for mass, momentum and energy of each phase while all the phases share a common pressure field.
- Eulerian averaging of the transport equations results in additional interaction between the phases.

Meshing

- An overset mesh is used to help an analysis that requires complex movement which cannot be simulated with a single mesh.
- An overset mesh combines two different meshing forms in a CFD simulation.
- Each mesh type is applied in different regions of the simulation such that the meshes overlap.
- An overset mesh is finished by enforcing continuity across the transition region between each mesh in the system.



General Equations

The Eulerian Multiphase Mixture models fluid phases by solving transport equations for mass, momentum, and energy for the mixture of phases as a whole rather than for each phase separately.

- To calculate the distribution of phases, the volume fraction transport equation is solved for each phase.
- For phases that are moving at different velocities, algebraic relations are used to compute the relative velocities.

The volume fractions are transported according to the following conservation equation:

$$\frac{\partial}{\partial t}\int_V \alpha_i dV + \int_A \alpha_i\, v_m \cdot da = \int_V \left(S_{u_i} - \frac{\alpha_i}{\rho_i}\frac{D_{\rho_i}}{D_t}\right) dV + \int_A \frac{\mu_i}{\sigma_i \rho_m}\nabla\alpha_i \cdot da - \int_V \frac{1}{\rho_i}\nabla\cdot\left(\alpha_i \rho_i V_{d,i}\right) dV$$

Where:

t is time
V is volume
α_i is the volume fraction of phase
v_m is the mass averaged velocity
a is the surface area vector
S_{u_i} is the user-defined source term for phase
ρ_i is the density of phase
μ_t is the turbulent dynamic viscosity
σ_t is the turbulent Schmidt number
$V_{d,i}$ is the diffusion velocity

The exact turbulent diffusion term is:

$$\frac{\mu_i}{\rho_i \sigma_i}\nabla Y_i = \frac{\mu_i}{\rho_i \sigma_i}\nabla\left(\frac{\rho_i}{\rho_m}\alpha_i\right) = \frac{\mu_i}{\rho_i \sigma_i}\left(\frac{\rho_i}{\rho_m}\nabla\alpha_i + \alpha_i \nabla\left(\frac{\rho_i}{\rho_m}\right)\right)$$

Where $Y_i = \frac{\alpha_i \rho_i}{\rho_m}$ is the mass fraction of phase i.

Boundary Conditions include a Slip Wall condition on the far-field computational domain top, bottom, and side walls, an inlet velocity wave, and hydrostatic pressure for the wave outlet.

General Equations

The mass of the spacecraft, center of mass and moment of Inertia is based on given data.

- During certain cases the translation and/or rotation of the vehicle were fixed especially for the simulation of the well deck.

Current, wind, wave heights, water depths, wave periods and wave lengths were all set to simulated specific wave conditions.

A fifth order approximation to the Stokes theory of waves is used to simulate the waves.

- The fifth order approximation more closely resembles a real wave than a wave that is generated by the first order method.
- The wave profile and the wave phase velocity depend on the water depth, wave height, and current.
- The fifth order VOF waves are based on work by Fenton[1].

The Ursell number[2] is defined as:

$$U_R = \frac{H\lambda^2}{d^3}$$

Where H, is the wave height, λ is the wavelength and d is the depth of the water.

- The Ursell number helps determine whether a wave's behavior can be approximated by a linear wave theory (i.e. Airy wave theory) or if nonlinear effects are significant.)
- This wave theory is generally valid for Ursell numbers less than approximately 5.

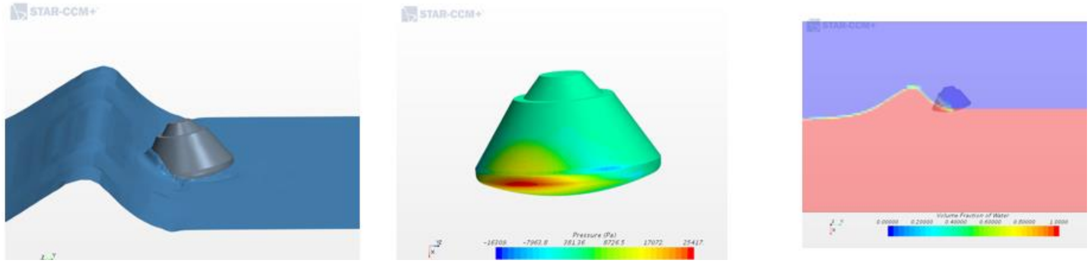
Analysis Results

Steep waves with short wave lengths and relatively small water depths are more likely to cause the capsule into an unstable condition and/or produce the highest backshell pressures.

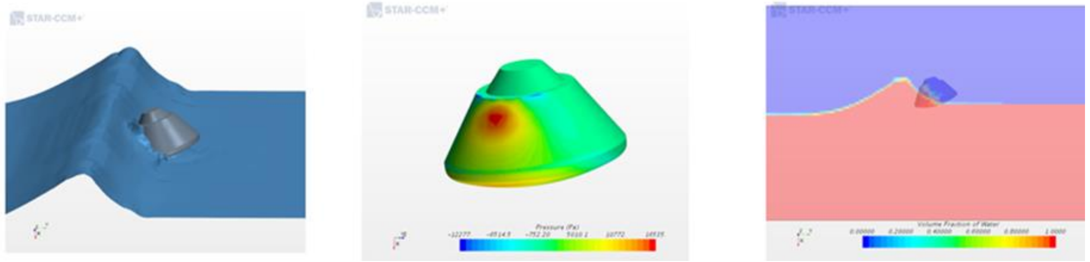
- A sea condition that breaks over the capsule will provide data on how the pressure vessel will react to repeated impact loads.
- This will help understand what sea conditions the capsule is able to operate in and the state of the pressure vessel.

The images of a breaking wave approaching the capsule is shown below.

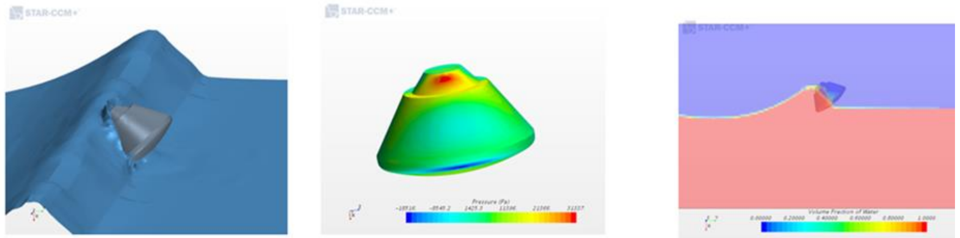
- The upper right image shows a contour plot of wave pressure on the surface of the vehicle as the wave spreads across the leading edge and around the curvature of the capsule outer shell.
- These pressures were used to predict the structure load on the outer panels of the vehicle.



Head Wave 0.15 sec



Head Wave 0.28 sec



Head Wave 0.47 sec

Analysis Results

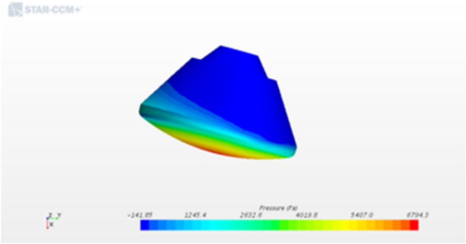
CFD models of different wave heights, test article under tow, and a simulated parachute water drop analysis are presented.

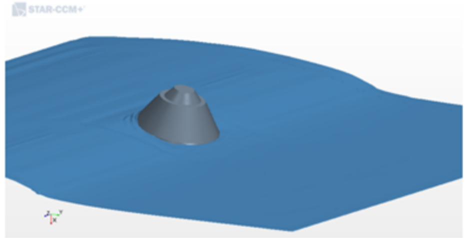
- The CFD results are shown for three maximum wave heights (4.0 m, 6.0 m, and 8.0 m) using a fifth-order wave model given the period of the wave.
- Each figure contains three images: an iso-surface plot (lower left), a wave induced pressure contour plot on the outer spacecraft shell (upper right) and a volume fraction of water plot (lower right).
- Additional images from a various number of studies performed on the vehicle are also shown.

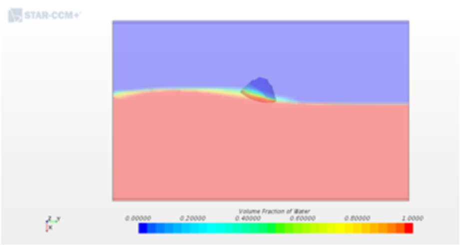
A 4 m wave approaching the capsule.

- The pressure contour plot shows the wave induced pressures on the capsule as the wave extends higher on the outer shell.

- Maximum Wave Height 4 m







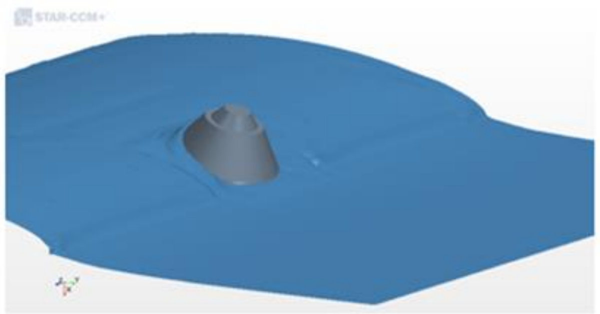
Iso-surface and Surface Pressure plots for 4 m Wave Heights

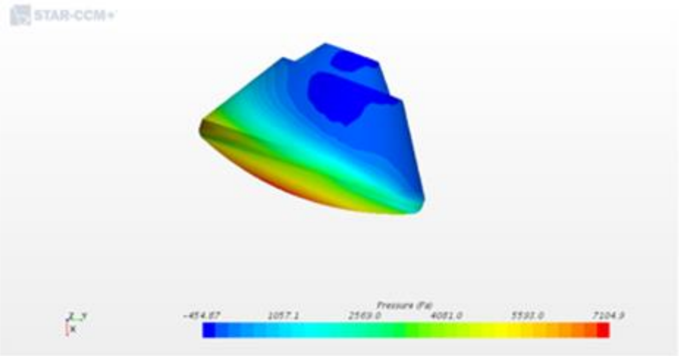
Analysis Results

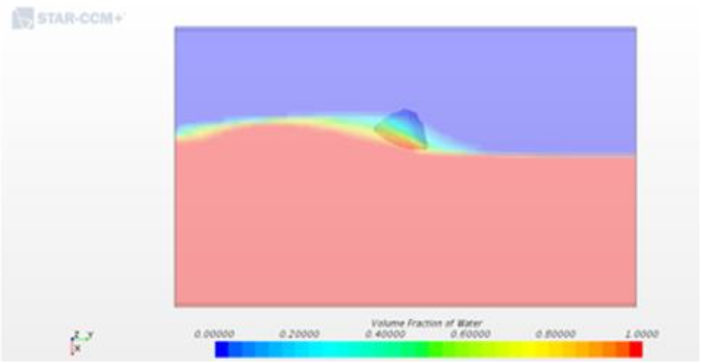
A 6 m wave approaching the capsule.

– The pressure contour plot shows the wave induced pressures on the capsule as the wave extends higher on the outer shell.

- Maximum Wave Height 6 m







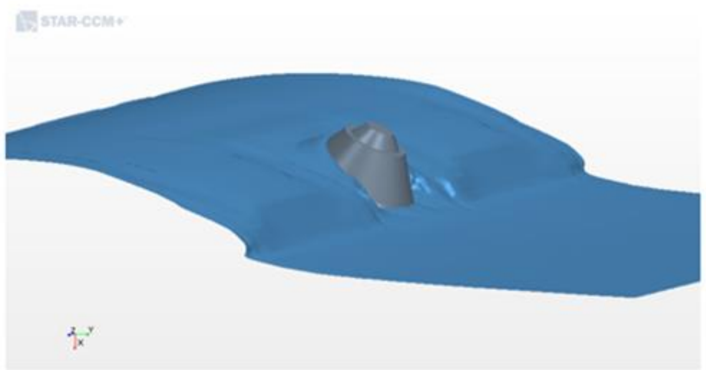
Iso-surface and Surface Pressure plots for 6 m Wave Heights

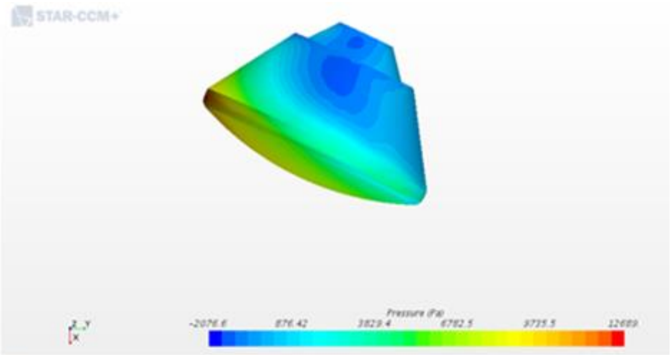
Analysis Results

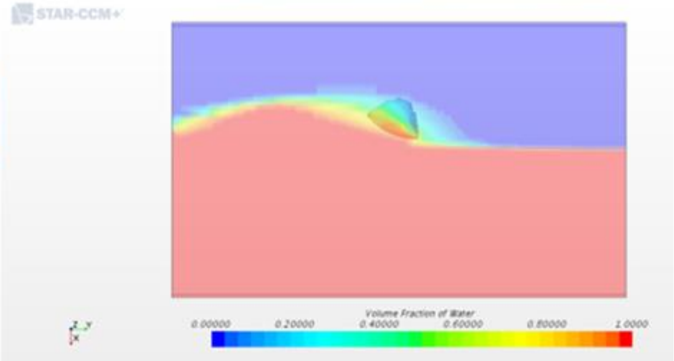
An 8 m wave approaching the capsule.

- The pressure contour plot shows the wave induced pressures on the capsule as the wave extends much higher on the outer shell.
- The pressure load is also increasing as it wraps around the structure.

- Maximum Wave Height 8 m





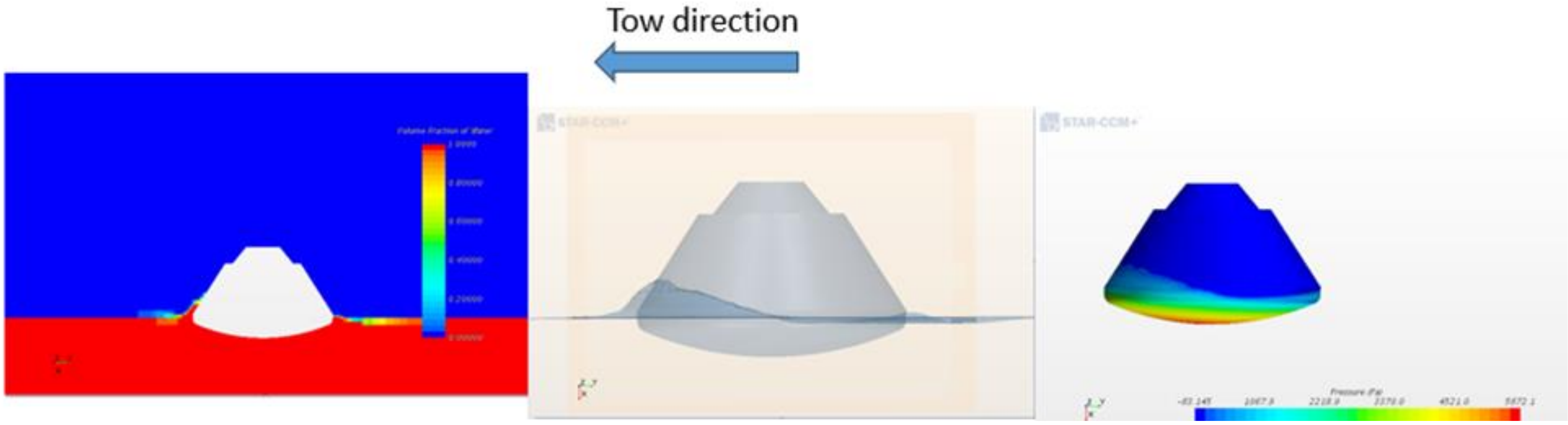


Iso-surface and Surface Pressure plots for 8 m Wave Heights

Analysis Results

The images below show the results of a simulation of a 6-knot tow on the outer shell of the hull.

- The spacecraft is simulated as being pulled from right to left in all three images.
  - The image on the left shows the volume fraction of water, the center image is an iso-surface plot (0.5), the image on the right is a pressure contour plot on the outer shell.
  - The plots all show the buildup of water on the leading edge of the hull as the vehicle is being towed.

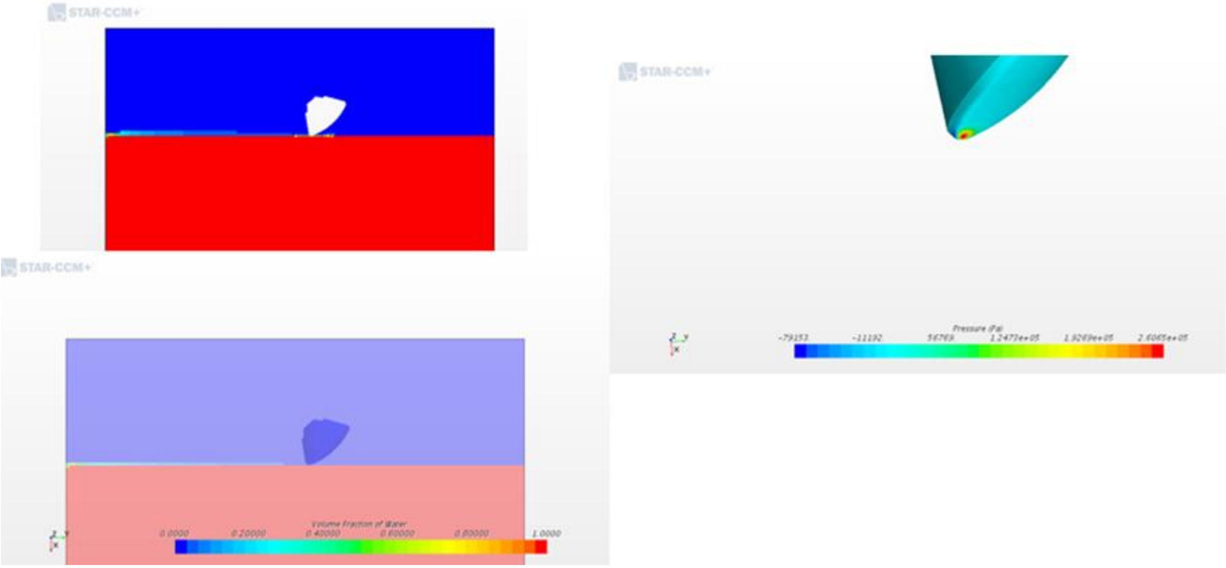


Simulation of a 6 knot tow

Analysis Results

The images below show a simulated parachute drop from an initial specific height and angle during its descent into the water.

- The left images shows a volume of fraction of water (red) contour plot of the spacecraft first impact at the water surface.
- The image on the right shows the initial pressure impact on the tip of the heat shield leading edge.

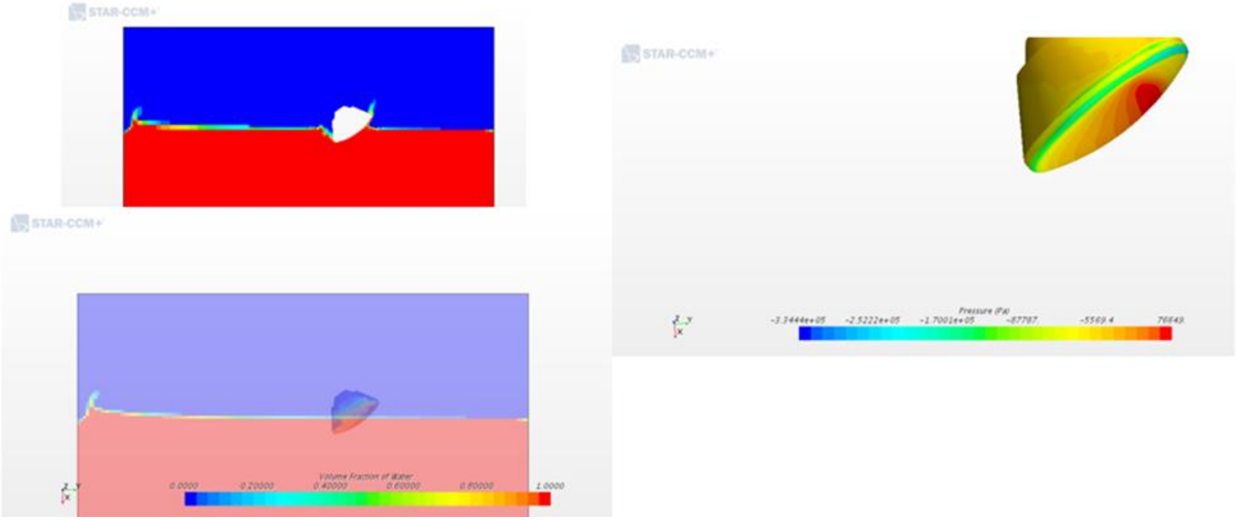


A Simulated Parachute Drop before Water Impact, Volume of Fraction Plot (Air Shown in Blue, Water Shown in Red).

Analysis Results

A simulated parachute drop from an initial specific height and angle during its descent into the water are shown.

- The left images shows a volume of fraction of water (red) contour plot of the spacecraft after first impact at the water surface.
- The image on the right shows the pressure impact on the bottom of the heat shield.



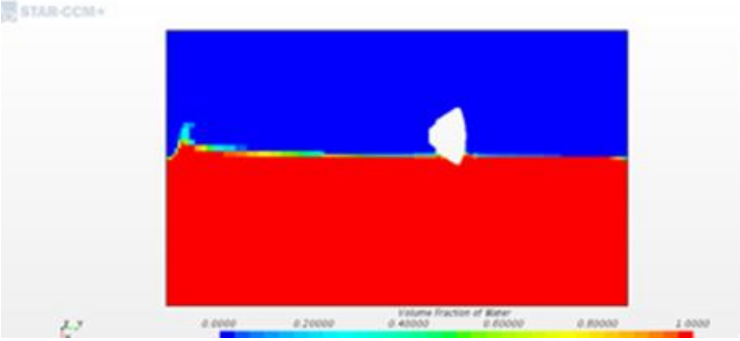
A Simulated Parachute Drop after Water Impact, Volume of Fraction (left image), Side View Pressure Contour (right image).

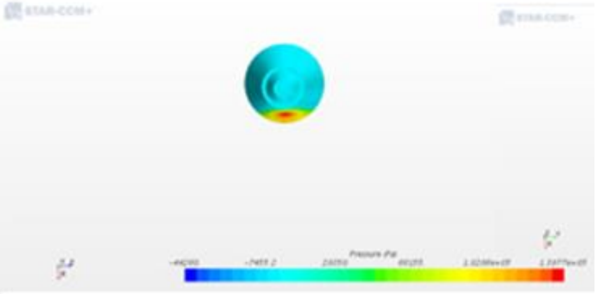
Analysis Results

The images below show a simulated parachute drop 0.278 and 0.303 seconds after the spacecraft impacts the water surface.

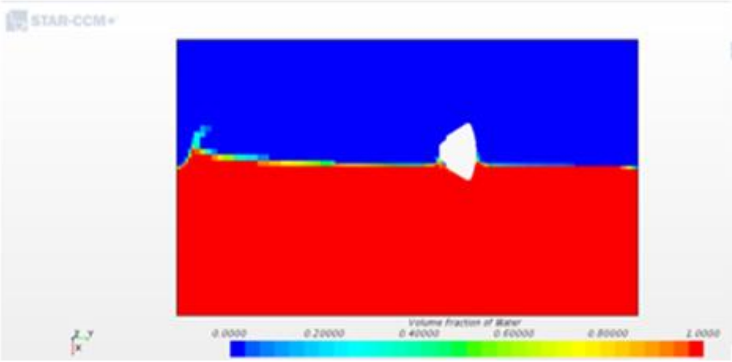
- The pressure contour plot on the right image shows the location of the peak pressures on the heat shield.

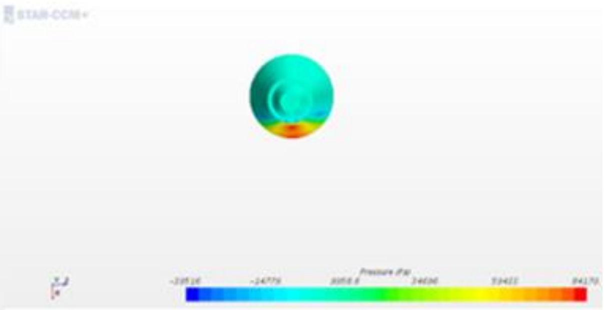
- Time 0.278 sec





- Time 0.303 sec

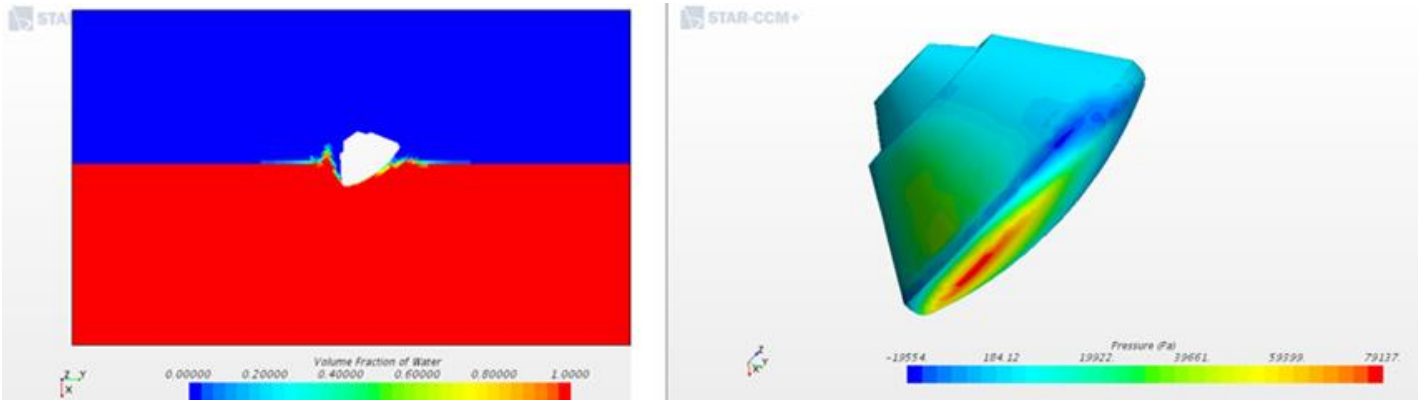




Analysis Results

The images below show a simulated parachute drop 0.5 seconds after the spacecraft impacts the water surface.

- The pressure contour plot on the right image shows the location of the peak pressures on the leading edge of the spacecraft panel.
- The area and magnitude of the pressures help determine the pressures for the structural analysis.

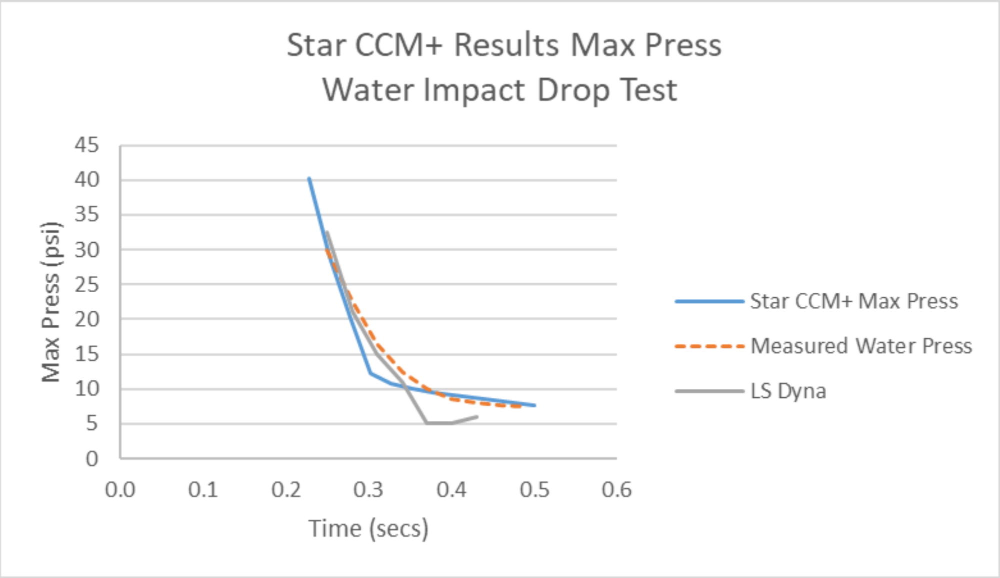


A Simulated Parachute Drop after Water Impact, Volume of Fraction (left image), Side View Pressure Contour (right image).

Analysis Results

The graph below compares the CFD maximum pressure results to measured pressure transducers on a training article during a water drop test conducted at NASA Langley Research Center's Hydro Impact Basin in 2016, and LS Dyna simulation results.

- The CFD results compared well to the measured pressure test data and LS Dyna results.
- The pressure data was initially higher at the time point when the test article made first contact with the water.
- This higher maximum pressure was at a different location from where the pressure transducers were located so it is possible that the higher pressure was not picked up by any of the pressure transducers.



Maximum Pressure on Outer Shell

Conclusion

This analysis describes the use of a three-dimensional, multi-phase, free surface transient computational fluid dynamics (CFD) model to simulate the wave impacts (pressure) on NASA's Artemis Crew Module Test Article (CMTA) outer shell under simulated sea conditions.

- The CFD analysis results helped determine pressure loads on the hull of a spacecraft test article that assessed the seakeeping/recovery effort for the Artemis lunar exploration program.
- The objectives of this analysis was to simulate the wave impacts (pressure) on the crew module's outer shell under simulated sea conditions.
- This analysis provided a record of seakeeping load cases so that follow-on structural analysis can use the pressure results to help determine the stress and distortion on the spacecraft training article hull.

References

1. https://www.nasa.gov/image-feature/orion-spacecraft-recovery-rehearsal-underway
2. Fenton, John D. 1985. “A Fifth-Order Stokes Theory for Steady Waves”, J. Waterway, Port, Coastal and Ocean Eng., 111 (2), pp. 216-234.
3. Det Norske Veritas. 2007. “Environmental Conditions and Environmental Loads”, Recommended Practice DNV-RP-C205, April 2007, pp. 27.
4. CD-adapco, 2019. *Star-CCM+ User Guide.*
5. Livermore Software Technology Corporation (LSTC), LS-DYNA, Livermore, CA, 2016.
6. https://www.nasa.gov/missions/nasa-trains-for-orion-water-recovery-ahead-of-artemis-ii-launch/

Drop Tests at NASA Langley's Hydro Impact Basin

Final Orion Spacecraft Drop Test, 20 May 2021
YouTube · SciNews · May 20, 2021
YouTube

- https://www.google.com/search?q=orion+Water+Impact+Tests&rlz=1C1GCEA_enUS1082US1082&oq=orion+Water+Impact+Tests+&gs_lcrp=EgZjaHJvbWUyBggAEEUYOdIBCTEzMjA2ajBqN6gCALACAA&sourceid=chrome&ie=UTF-8#fpstate=ive&vld=cid:7137692d,vid:DhN2zEvKx-4,st:0
- https://www.youtube.com/watch?v=Raou4AV_B-E
- https://www.youtube.com/watch?v=wZEiv5vq1yo&list=PLOuu0akB6CSi_kznNkDnjx4lT7Pa5scWK&index=2

Questions

Thank you