Hydraulic and Kinematic System Model for TBCC Dynamic Simulation

Eric J. Gamble* and Dan Haid†
SPIRITECH Advanced Products, Inc, Tequesta, Florida

A dynamic simulator is being developed to demonstrate all modes of operation, including mode transition, for a Turbine-Based Combined Cycle (TBCC) propulsion system. The High Mach Transient Engine Cycle Code (HiTECC) is a highly integrated simulation tool comprised of models for each of the TBCC systems whose performance and controllability affect the thrust and operability of the propulsion system. The reported work details the development of the Hydraulic and Kinematic System models conducted in the second year of a multiyear effort to develop a dynamic TBCC simulator. Once completed, this model will significantly extend the state-of-the-art for all TBCC modes of operation by providing a numerical simulation of the systems, interactions, and transient responses affecting the ability of the propulsion system to transition from turbine-based to ramjet/scramjet-based propulsion.

I. Nomenclature

A	area
F	Aerodynamic Load
I	moment of inertia
M	Moment
p	pressure
R	Actuator Reaction Load
t	time (sec)
θ	angle (°)
HS	high speed flow path
LS	low speed flow path
CG	center of gravity
CS	coordinate system

Subscripts

0	freestream conditions
i	spatial index
atm	atmospheric condition
s	static conditions
t	total conditions
x	axial direction
y	vertical direction

II. Introduction

THE need for the National Aeronautics and Space Administration (NASA) Aeronautics Research Mission Directorate (ARMD) Hypersonic Project is based on the fact that all access to earth or planetary orbit and all entry into earth's atmosphere or any heavenly body with an atmosphere from orbit (or super orbital velocities) require flight through the hypersonic regime. The hypersonic flight regime often proves to be the design driver for most of the vehicle's systems, subsystems, and components. If the United States wishes to continue to advance its capabilities for space access, entry, and high-speed flight within any atmosphere, improved understanding of the hypersonic flight regime and development of improved technologies to withstand and/or take advantage of this environment are required.

A critical element of NASA's hypersonics research is the development of combined cycle propulsion systems, including rocket-based combined cycles (RBCC) and turbine-based combined cycles (TBCC). Based on the Next Generation Launch Technology (NGLT), TBCC, Two-Stage-to-Orbit (TSTO), National AeroSpace Plane (NASP), and High Speed Propulsion Assessment (HiSPA) studies, a turbofan and ramjet variable cycle engine is best suited

* Vice President, SPIRITECH Advanced Products, Inc., member AIAA
† Aerodynamic Engineer, SPIRITECH Advanced Products, Inc., member AIAA

to satisfy the access-to-space mission requirements by maximizing thrust-to-weight ratio while minimizing frontal area and maintaining high performance and operability over a wide operating range. The TBCC Dynamic Simulation Model Development Program discussed in this paper advances the technology readiness level of TBCC systems by developing the simulation and controls software to model all modes of operation over its mission, including mode transition from gas turbine to dual-mode scramjet propulsion, a requirement before controlled wind tunnel testing or flight testing can be accomplished through this region of operation. Within this program, modeling tools are being developed from fundamental physics and are being integrated into a comprehensive dynamic simulation tool to determine the transient performance, providing actual event durations to properly configure the propulsion system control logic. Work accomplished to date includes the development of the Propulsion System[1], the Thermal Management and Fuel System[2], and the Hydraulic and Kinematic System discussed here.

III. Technical Discussion

A. System Summary

The Hydraulic and Kinematics System simulates the variable geometry features of the inlet and nozzle for both the low-speed and high-speed flow paths. The system includes a flow model for determining the dynamic response of the hydraulic fluid, a kinematic model for the low-speed and high-speed inlet cowls and nozzle flaps, and models for the power storage and generation for pumping the hydraulic fluid.

The system, summarized in Figure 1, is divided into four sub-systems. The first, the flow sub-system, models the fluid and energy flow through actuators, valves, and other hydraulic components. The second, the kinematics subsystem, models the loads and energy transfer through body (links) and joint components. The third, the power sub-system, models the flow of power from the battery, motor, and pump components. Finally, the storage sub-system models fluid flow to and from the storage tank and other plumbing components associated with it.

The Hydraulic and Kinematic System is required to simulate fluid flow and energy transfer throughout the hydraulic system and mechanical loads and energy transfer through the kinematics system. The hydraulics are modeled with a one-dimensional incompressible flow solver. The kinematics are modeled with a dynamic rigid body model. The system requires the user to input the arrangement of the components, their specifications, and the hydraulic fluid type. Output from the system includes data from the individual components that the user can utilize to adjust the system sizing.

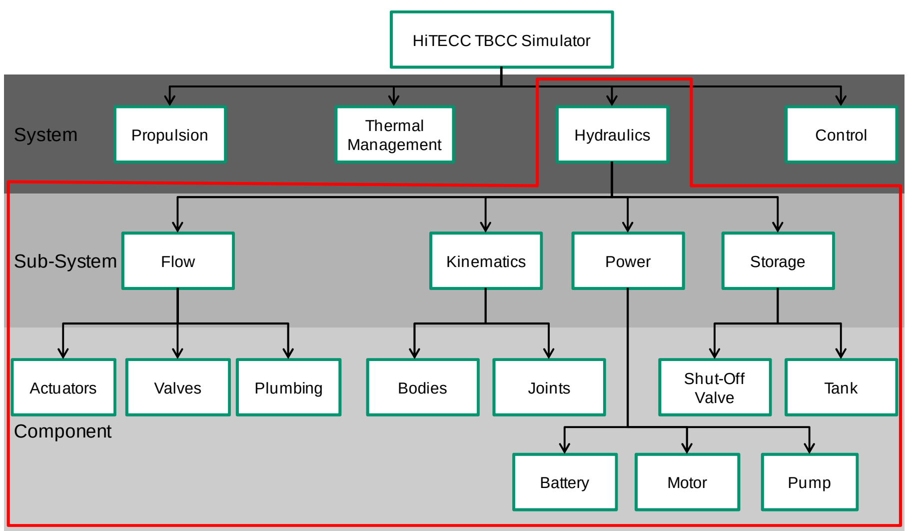


Figure 1. HiTECC TBCC Simulator Organization.

B. System Design

1. *Environment*

The Hydraulic and Kinematic System is modeled in the Simscape™ environment. This environment was selected for a number of reasons. Other simulator systems use Simscape™ and/or Simulink™, and by keeping a common environment, interfaces can be made to be simple and computationally efficient. A common solver selected from Simulink™'s list of proven solvers can be used to reduce risk. Finally, the Graphical User Interface is common among systems and familiar to the user, allowing the user to understand the interfaces between systems, sub-systems, and components and to make changes to them as needed.

The system requires physical models in the hydraulic and mechanical domains. Simscape™ includes basic hydraulic and mechanical components found in the foundation library that will be used throughout the Hydraulics and Kinematics System. HiTECC's higher level component requirements, such as actuators, pumps, and bodies, use the Simscape™ vertical products, SimHydraulics™ and SimMechanics™.

2. *Hydraulic Model*

The hydraulic model is divided into three subsystems that classify components by storage, power, or delivery functions. The components within each subsystem required for the HiTECC simulation have been developed within the Simscape™ environment. Several components were taken from the Simscape™ foundation library while others were taken from the higher level SimHydraulics™ library. Those not found in either the foundation or SimHydraulics™ library were built from Simscape™ components. Examples of these components include the battery and motor from the power subsystem.

The Storage Subsystem components include a tank and shutoff valve components. Both the Reservoir and 2-Way Directional Valve components have been selected from the SimHydraulics™ library to act as a tank and shutoff valve, respectively. The Reservoir block represents a pressurized hydraulics reservoir in which the pressure remains constant regardless of volume change. The block accounts for pressure loss in the return line that can be caused by a filter, fittings, or some other local resistance. The 2-Way Directional Valve block simulates a valve as a data sheet-based model. The controlling parameters are the maximum area of the valve opening and the control member stroke.

The Power Subsystem components include a battery, motor, and pump. The battery and motor models were constructed with Simscape™ blocks and arranged to model a DC power system. The battery is modeled as an ideal voltage source and is connected to the motor model. The DC motor model has the ability to cross domains from electrical to mechanical with the Rotational Electromechanical Converter block. The energy consumed by the motor, from the battery, is recorded throughout the simulation to determine the system's power requirements. To increase modeling accuracy, inertia is added to the motor model to simulate the mass of the rotating components. A Fixed-Displacement pump from the SimHydraulics™ library is included and attached to the motor model. The input for the pump is representative of a data sheet-based model. The key parameters required to parameterize the block are the pump displacement, volumetric and total efficiencies, nominal pressure, viscosity, and angular velocity.

The Delivery Subsystem components include actuators, plumbing, and control valves. The actuator selected is from the SimHydraulics™ library and represents a double–acting hydraulic cylinder. It is built from several Simscape™ blocks and includes a translational Hydro-Mechanical Converter to cross from the fluid to mechanical domain. The actuator's rod motion is limited with mechanical translational hard stops for which the stiffness and damping can be defined. Plumbing in the Delivery Subsystem is modeled with a Hydraulic Pipeline block that accounts for circular and non-circular cross-sections, friction losses along the pipe length, and fluid compressibility. The control valve for the double-acting hydraulic cylinder is a 4-way directional valve which is modeled as a data sheet-based model. To parameterize the block, the maximum area and control member stroke are entered.

3. *Kinematic Models*

The MathWorks™ modeling software SimMechanics™ has been used to extend Simscape™ to include modeling of three-dimensional mechanical systems within the Simulink™ environment. This multi-body simulation tool is used to build models composed of bodies, joints, constraints, and force elements that reflect the structure of the system. The mass, inertia, and center of gravity are all captured within this environment. The visual block model method of SimMechanics™ allows for intuitive interpretation of the models without the need to derive and program equations. The mechanical systems modeled in the simulator are the variable geometry inlet and the variable geometry nozzle.

The inlet component couples HiTECC's existing geometry routines, which provide geometry data to the performance model, with the SimMechanics™ kinematic system to model the inlet shown in Figure 2. Integration of these allows for actuated control of the inlet's various components. In total, three components have been actuated, including the high-speed cowl lip, low-speed cowl lip, and variable geometry ramps.

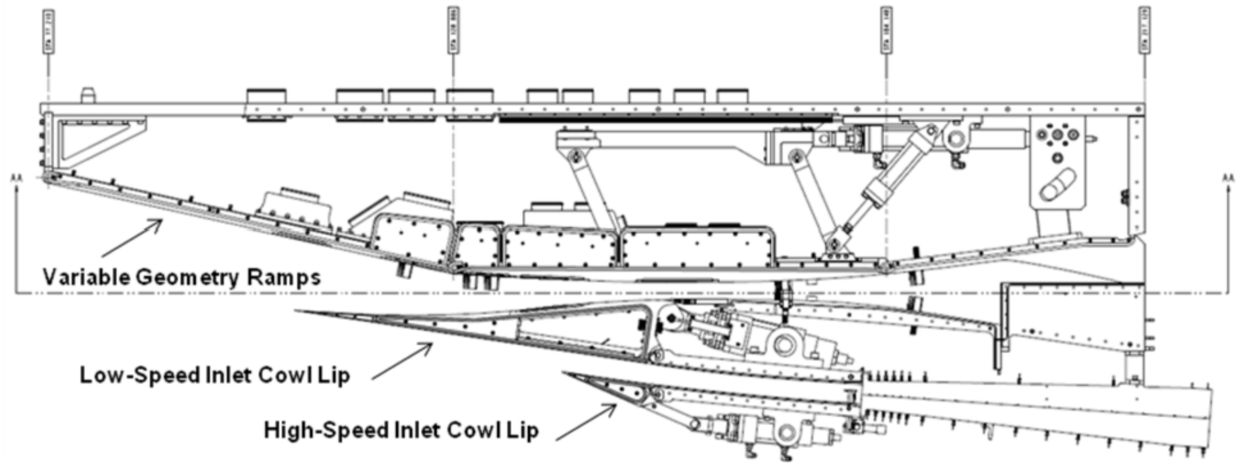


Figure 2. Inlet Actuated Components

High and low-speed cowl lips rotate about their respective body connection points, found in Figure 3, with the flap structure directly connected to the driving actuators. These actuators are positioned in the cavities aft of the flaps and are supported by rotational joints grounded to the aircraft.

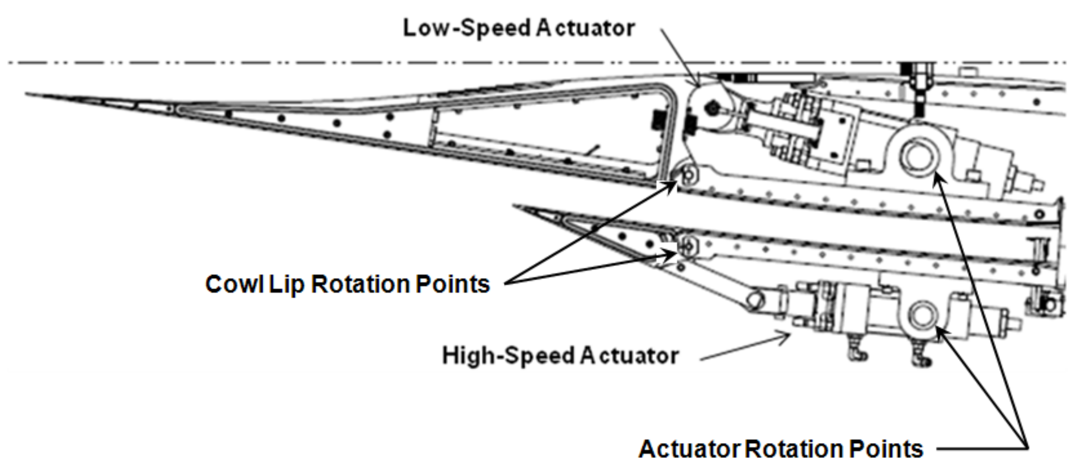


Figure 3. Low and High Speed Cowl Lip Actuators

The kinematic mechanism for the variable geometry ramps follows a four bar parallelogram linkage design, as shown in Figure 4. The ramps are actuated with one actuator positioned within the cavity adjacent to the ramps themselves. Additional actuators can be added, as required, simply by copying the actuator model and attaching it at the desired location. A sample of the SimMechanics^{TM} block diagram model of the low speed cowl is shown in Figure 5 and includes data interfaces with the Propulsion System.

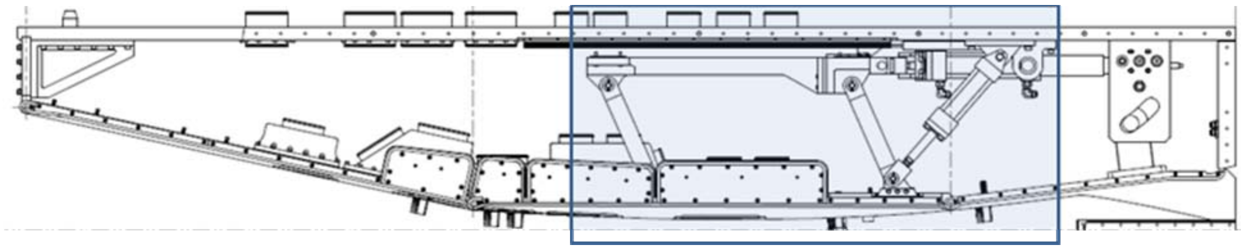


Figure 4. Variable Geometry Ramp Linkage Design

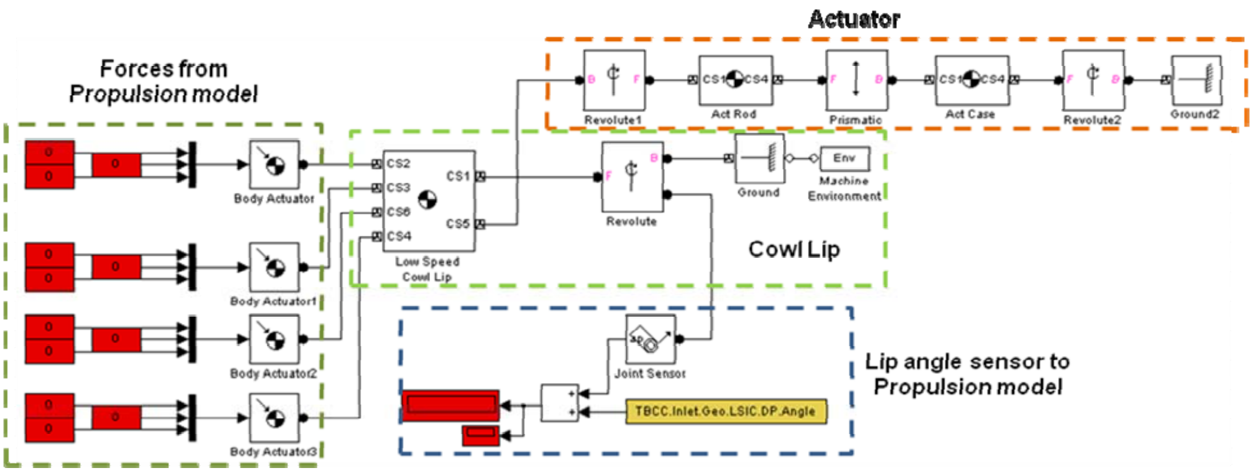


Figure 5. SimMechanics™ Low Speed Cowl Model

Similar to the inlet subsystem, the nozzle geometry and the kinematics subsystem are integrated between SimMechanics™ and the existing MATLAB™/Simulink™ geometry routines. Control of the nozzle has been accomplished with the use of four actuators – three with accompanying bell crank mechanisms, and one with a sliding linkage. The low and high-speed convergent flaps and high-speed lower flap are linked directly to their individual bell crank mechanisms while the low-speed divergent flap is driven with a sliding link mechanism. The nozzle arrangement, shown in Figure 6, allows for the high-speed divergent flap to be positioned without the use of a directly linked actuator.

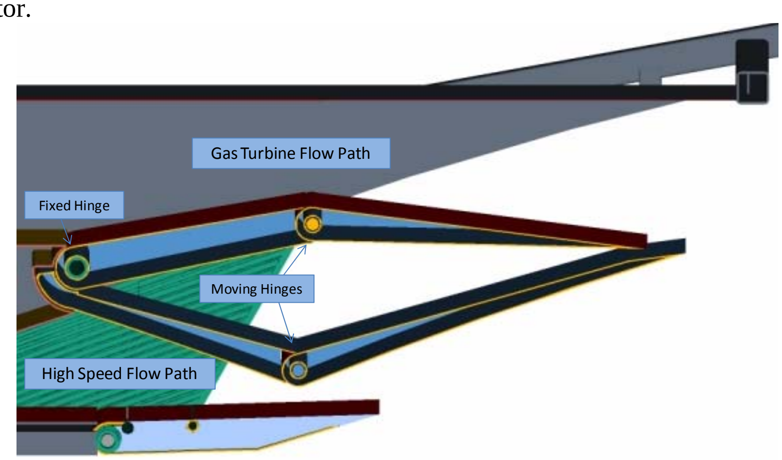


Figure 6. Nozzle Flap Arrangement

A partial snapshot of the SimMechanics™ block diagram model of the nozzle is included in Figure 7 and includes multiple data interfaces with the Propulsion System and a custom interface joint between the low-speed divergent flap and the high-speed divergent flap.

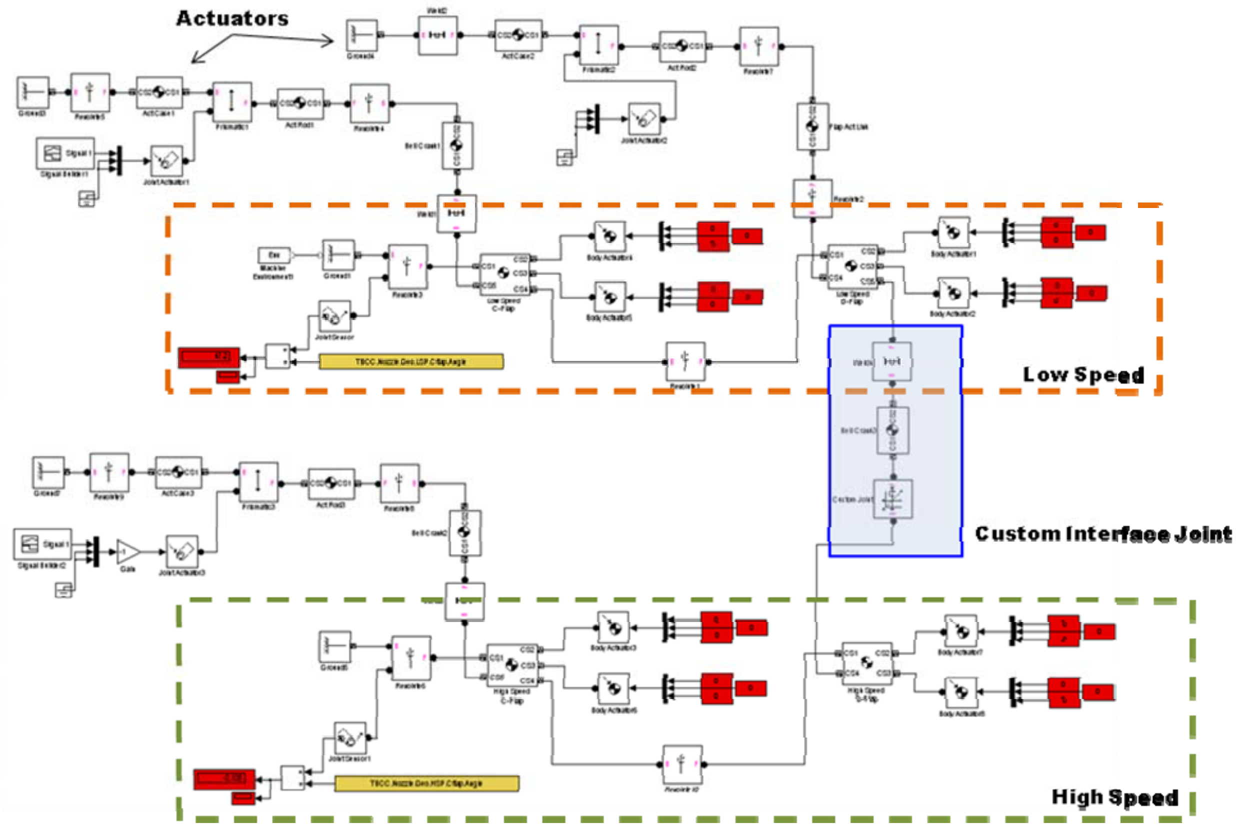
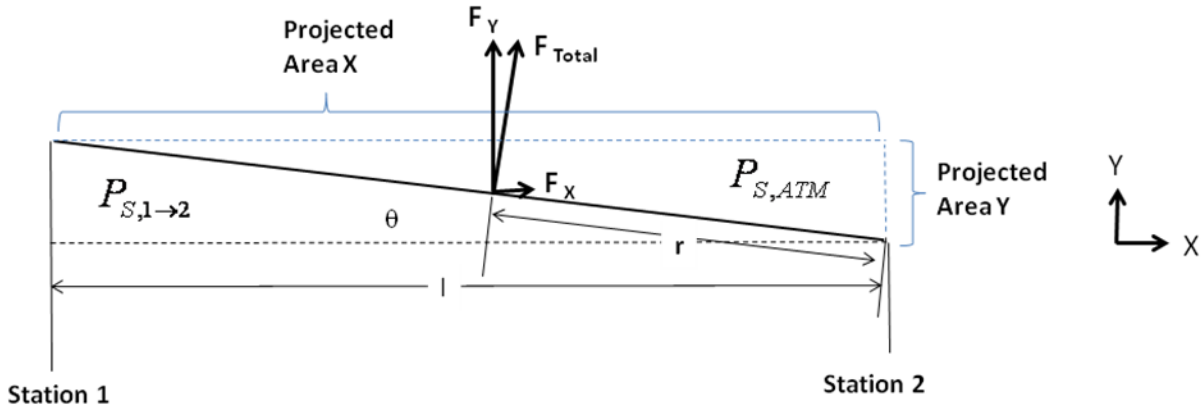


Figure 7. SimMechanics™ Nozzle Model

4. Aerodynamic Load Integration

The method for calculating aerodynamic loading on both the internal and external flow path surfaces has been implemented for the inlet and nozzle models. The method uses the flow path pressures determined at each of the geometric stations to calculate the average pressure acting on the flow path side of each of the ramp, cowl, and flap segments. The load is then calculated using the pressure differential ($\Delta p=p_{s,gaspath}- p_{s,backside}$) acting on the segment, along with the appropriate geometric projected areas, A. The calculated load components, F_x and F_y of the applied force vector, F_{total}, are shown in Figure 8. The calculated applied forces along each segment are output to the Hydraulic and Kinematic System, which statically determines the reaction, R, along each of the actuators and grounding locations.



$$\Delta P = P_{S,1\to 2} - P_{S,ATM} \qquad F_{X,1\to 2} = \Delta P A_Y$$

$$P_{S,1\to 2} = \frac{P_{S,1} + P_{S,2}}{2} \qquad F_{Y,1\to 2} = \Delta P A_X$$

Figure 8: Force Calculations using Projected Areas

Preference was given to this method versus other more complex approaches in order to maintain reasonable accuracy and minimize run-time of the overall code. This approach has been implemented and is being evaluated using CFD-generated pressure distributions. Preliminary investigations have shown that the error associated with this simple integration method applying a single load at the flap midpoint is reduced by 70% by adding a second segment, and therefore, a second load, to each flap. Additional accuracy improvements may be provided by adding more segments to each flap, as illustrated in Figure 9. This increased segmenting provides higher fidelity to the load calculation at the expense of additional computational run times.

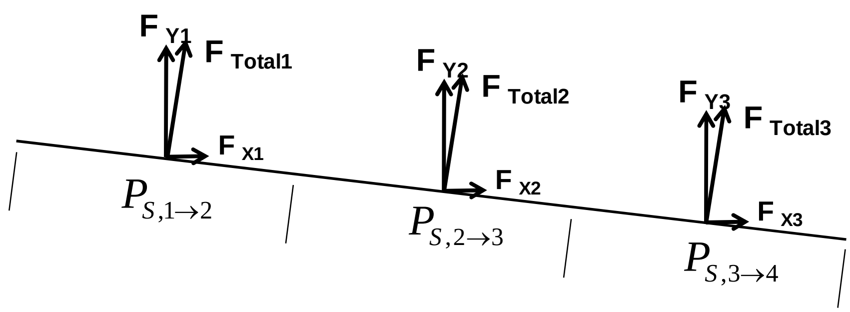


Figure 9: Additional Segment Divisions for Higher Fidelity Pressures

5. Load Modeling

A method for calculating the aerodynamic loadings on both the internal and external flow path surfaces has been implemented for the inlet and nozzle models. Aerodynamic flow conditions were used to determine the applied forces, *F*, on each of the ramp, cowl, and flap surfaces. These applied forces were then used to determine the reaction load, *R*, imparted to the actuators. Forces and moments for the inlet and nozzle are represented in Figure 10 and Figure 11, respectively.

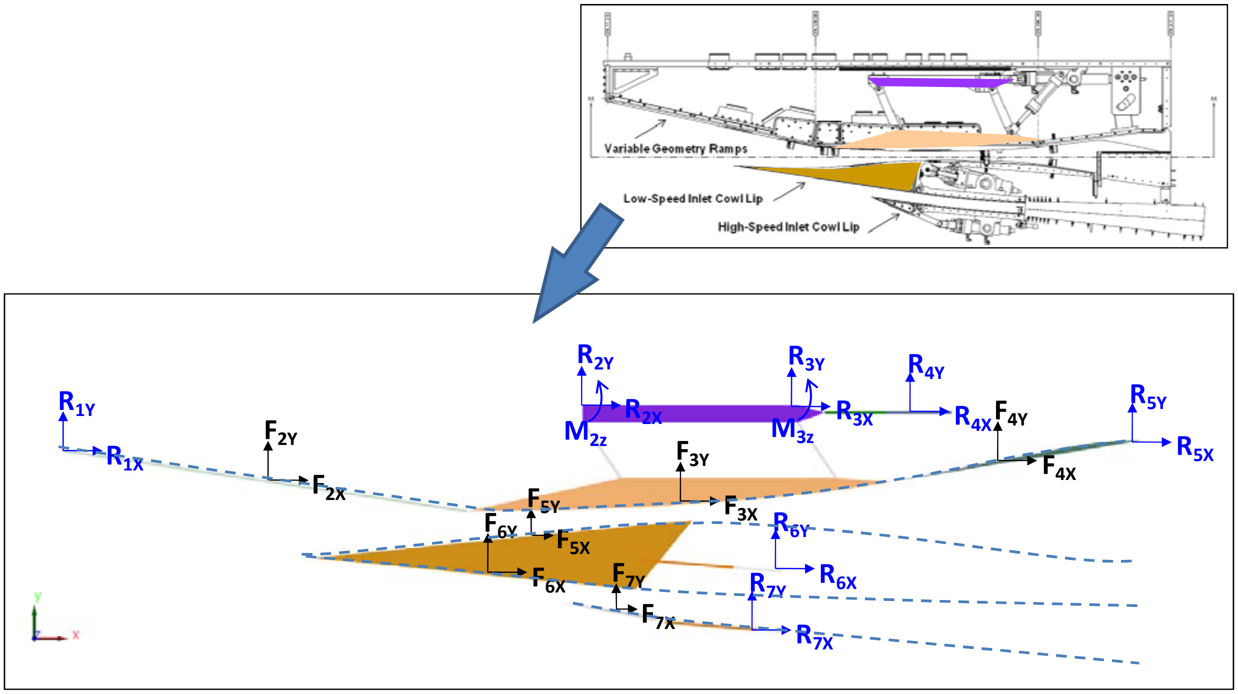


Figure 10: Inlet Forces and Moments

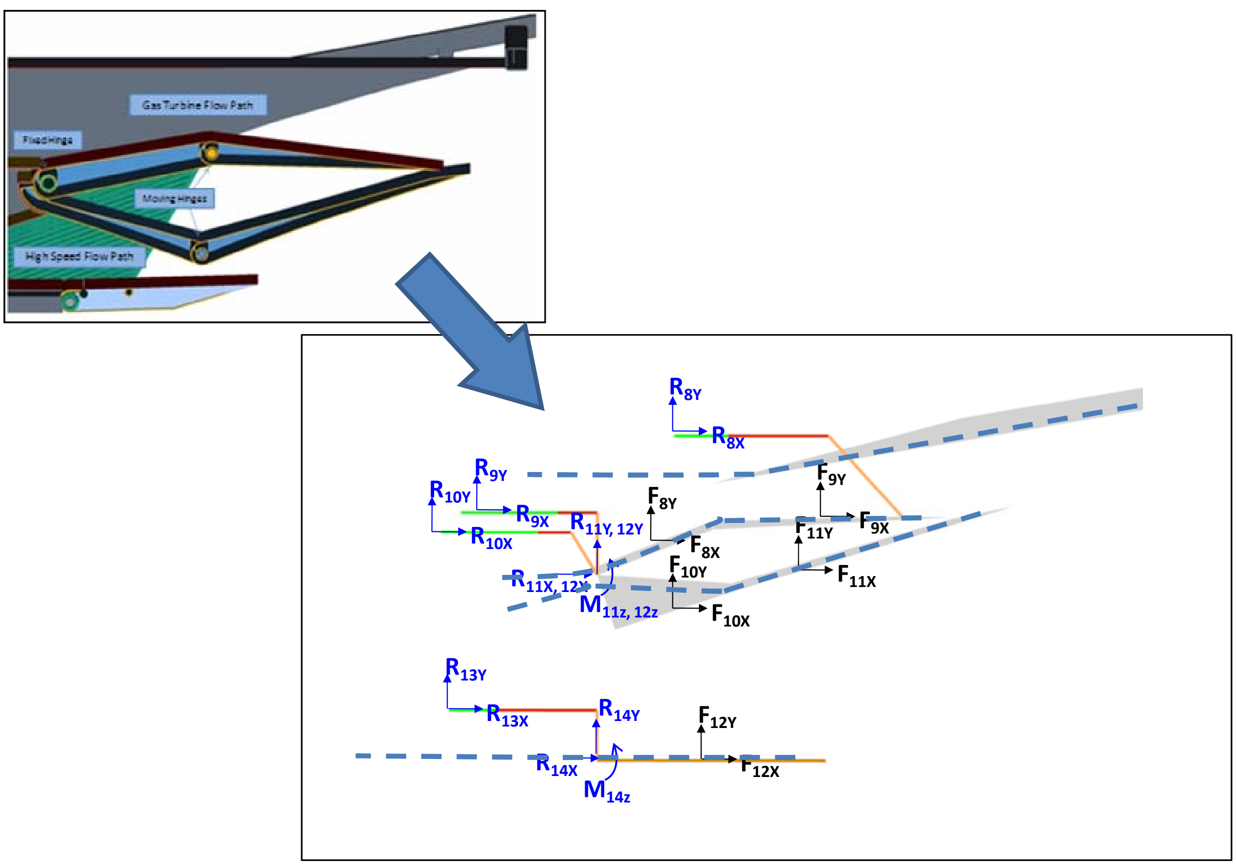


Figure 11: Nozzle Forces and Moments

IV. Model Verification

A. Inlet Kinematic Model Verification

A finite element model (FEM) was created and analyzed using ANSYS to validate the HiTECC kinematic model. The geometry used in the models was based on the design configuration run for a Mach 2.5 condition. The inlet FEM model is shown in Figure 12.

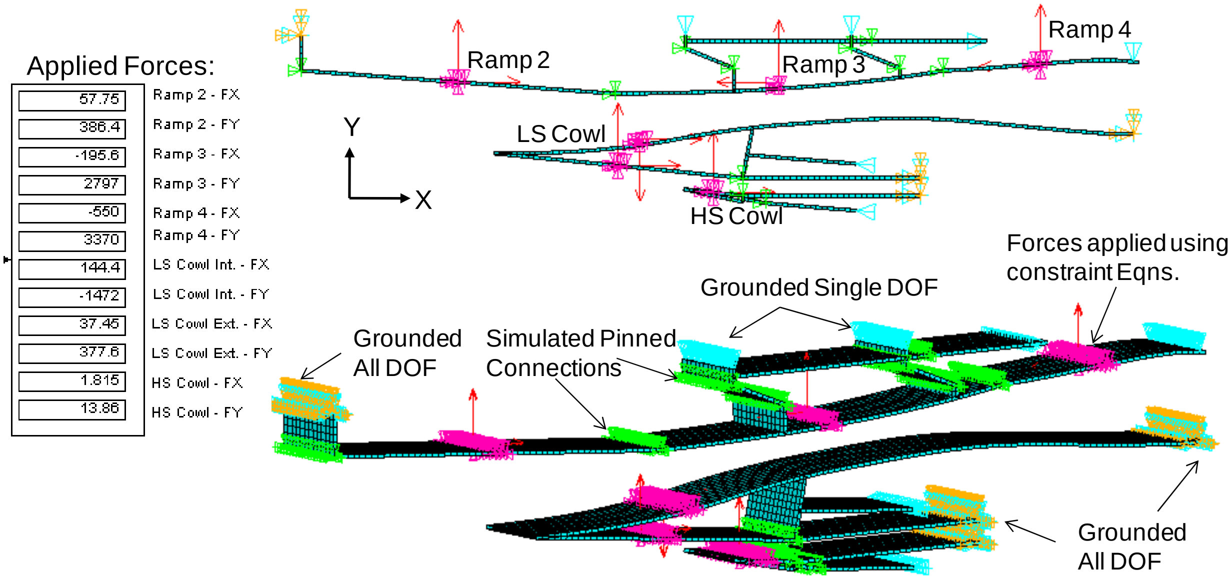
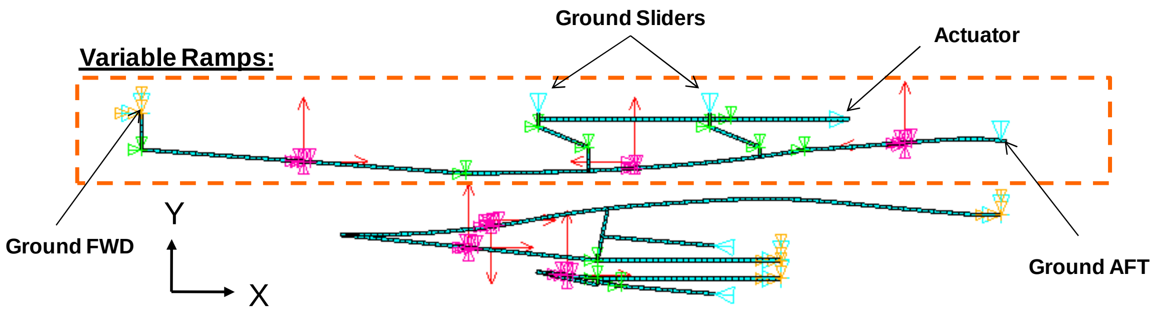


Figure 12: Inlet Verification - FEM with Applied Forces

The model consists of variable ramps (Ramp 2, 3, and 4), a low-speed cowl, and a high-speed cowl. The model was constructed using SHELL93 elements with approximate part thicknesses applied using real constants. Forces were applied to the FEM to match what was generated in the HiTECC model. The FEM constraints also matched what was used in the HiTECC kinematics model in order to simulate the physical system. Model constraints included fixed nodal degree of freedom (DOF) to simulate grounding locations, and nodal couplings were used to simulate the pinned joints. The forces used in the HiTECC model were transferred to the FEM using constraint equations. These equations distributed the applied forces onto each of the inlet ramp and cowl surfaces. The analysis models were then solved and resulting reactions along each of the constraint locations were used for the comparisons with the HiTECC results. FEA results for the variable ramps are shown below in Figure 13. Force reactions calculated along the grounding locations and actuator shows a close comparison between the HiTECC and FEM models. The largest difference of 0.2% occurred along the aft grounding location

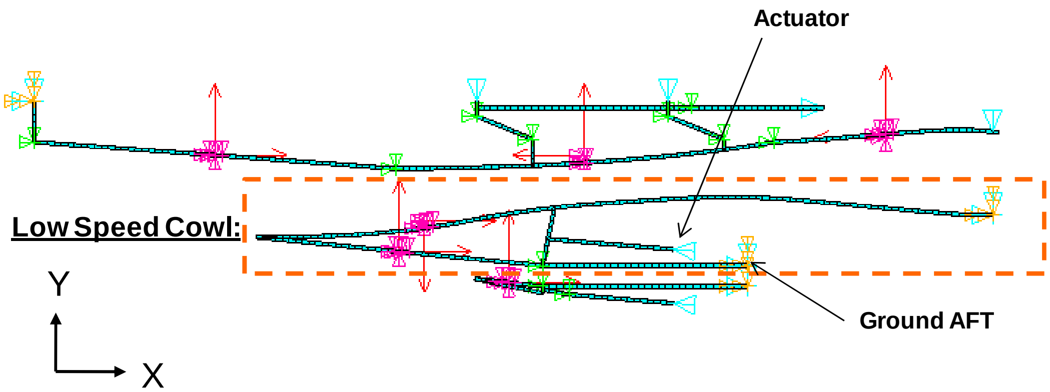


				Variable Ramps:				
Actuator					Ground FWD			
Reaction	HiTECC	Ansys	% Diff.		Reaction	HiTECC	Ansys	% Diff.
RX	13997	13997	0.00%		RX	-13310	-13309	0.01%
RY	n/a	n/a	n/a		RY	782	782	0.05%
RZ	n/a	n/a	n/a		RZ	n/a	n/a	n/a
Ground AFT					Ground Sliders			
Reaction	HiTECC	Ansys	% Diff.		Reaction	HiTECC	Ansys	% Diff.
RX	n/a	n/a	n/a		RX	n/a	n/a	n/a
RY	-1697	-1693	0.22%		RY	-5641	-5642	0.01%
RZ	n/a	n/a	n/a		RZ	n/a	n/a	n/a

Figure 13: Inlet Variable Ramps - Force Results Comparison (HiTECC vs. ANSYS FEM)

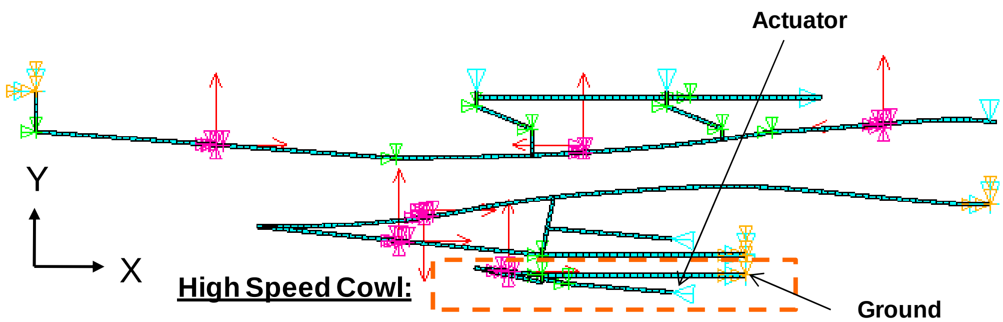
.

Force reaction results for the low-speed and high-speed cowls are shown below in Figure 14 and Figure 15. All locations showed a close comparison between the FEM and the HiTECC code with differences below 0.2%.



				LS Cowl:				
Actuator					Ground			
Reaction	HiTECC	Ansys	% Diff.		Reaction	HiTECC	Ansys	% Diff.
RX	4173.0	4178.9	0.14%		RX	-4355.0	-4360.8	0.13%
RY	-337.1	-337.5	0.12%		RY	1432.0	1431.9	0.01%
RZ	n/a	n/a	n/a		RZ	n/a	n/a	n/a

Figure 14: Inlet Low Speed Cowl - Force Results Comparison (HiTECC vs. ANSYS FEM)



				HS Cowl:				
	Actuator					Ground		
Reaction	HiTECC	Ansys	% Diff.		Reaction	HiTECC	Ansys	% Diff.
RX	76.38	76.43	0.07%		RX	-78.20	-78.20	0.00%
RY	-6.27	-6.28	0.08%		RY	-7.59	-7.60	0.09%
RZ	n/a	n/a	n/a		RZ	n/a	n/a	n/a

Figure 15: Inlet High-Speed Cowl - Force Results Comparison (HiTECC vs. ANSYS FEM)

B. Nozzle Kinematic Model Verification:

A nozzle FEM was also created and analyzed using ANSYS to validate the nozzle kinematic model in HiTECC. The modeling approach used for the nozzle was similar to that used in the inlet FEM. The model consists of a low-speed convergent flap (LS C-Flap), low-speed divergent flap (LS D-Flap), high-speed convergent flap (HS C-Flap), high-speed divergent flap (HS D-Flap), and high-speed lower flap (HS L-Flap). The model was constructed using SHELL93 elements with approximate part thicknesses applied using constants. Figure 16 shows the model along with the constraints and applied loads.

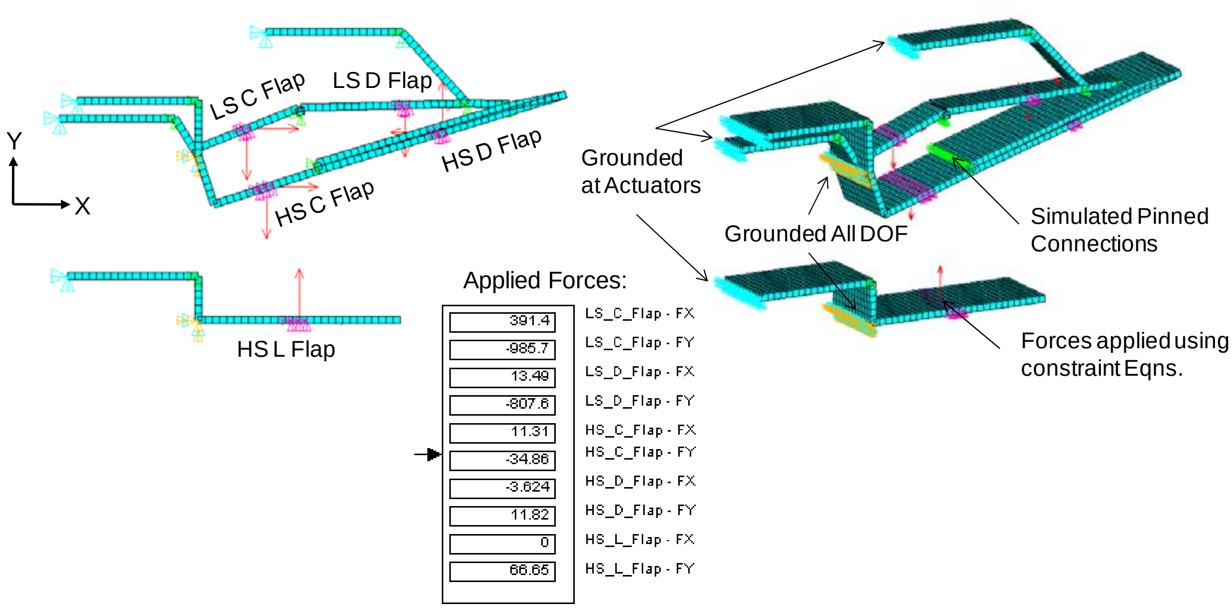
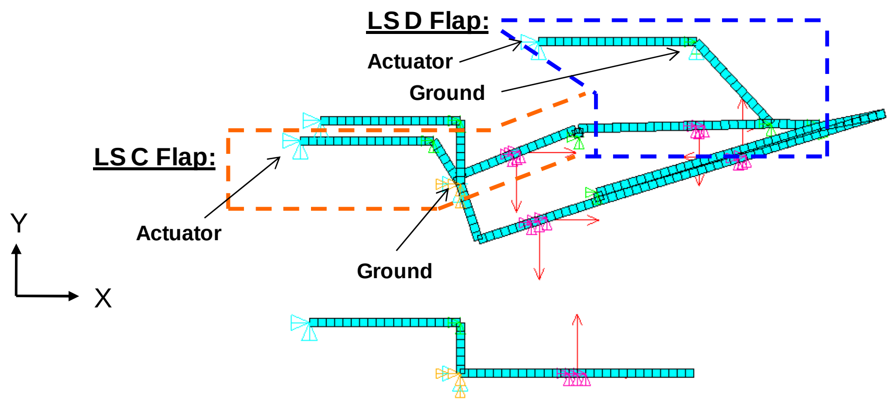


Figure 16: Nozzle Verification - FEM with Applied Forces

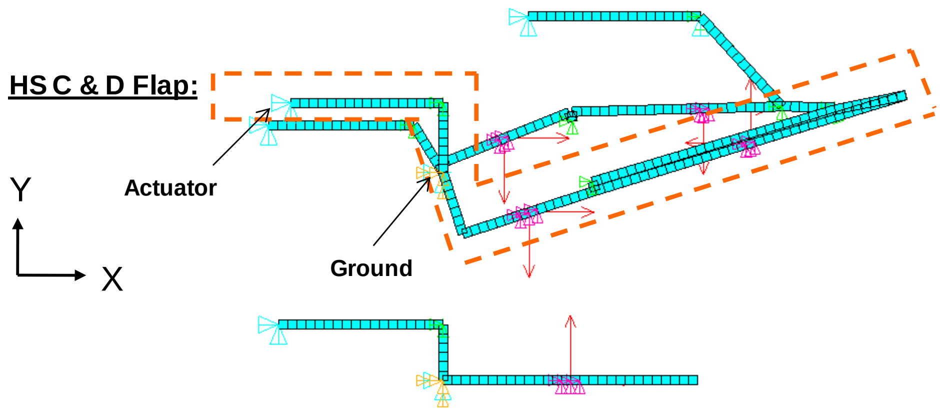
Results for the low-speed flaps are shown in Figure 17. Force reactions taken at the grounding locations and actuators in the FEM show a close comparison with the forces calculated in the HiTECC code with differences below 0.1%.



				LS C Flap:				
Actuator					Ground			
Reaction	HiTECC	Ansys	% Diff		Reaction	HiTECC	Ansys	% Diff
RX	-2020.4	-2021.0	0.03%		RX	2044.0	2043.2	0.04%
RY	n/a	n/a	n/a		RY	1320.0	1319.8	0.02%
RZ	n/a	n/a	n/a		RZ	n/a	n/a	n/a
				LS D Flap:				
Actuator					Ground			
Reaction	HiTECC	Ansys	% Diff		Reaction	HiTECC	Ansys	% Diff
RX	-425.0	-424.9	0.02%		RX	n/a	n/a	n/a
RY	n/a	n/a	n/a		RY	466.0	465.9	0.02%
RZ	n/a	n/a	n/a		RZ	n/a	n/a	n/a

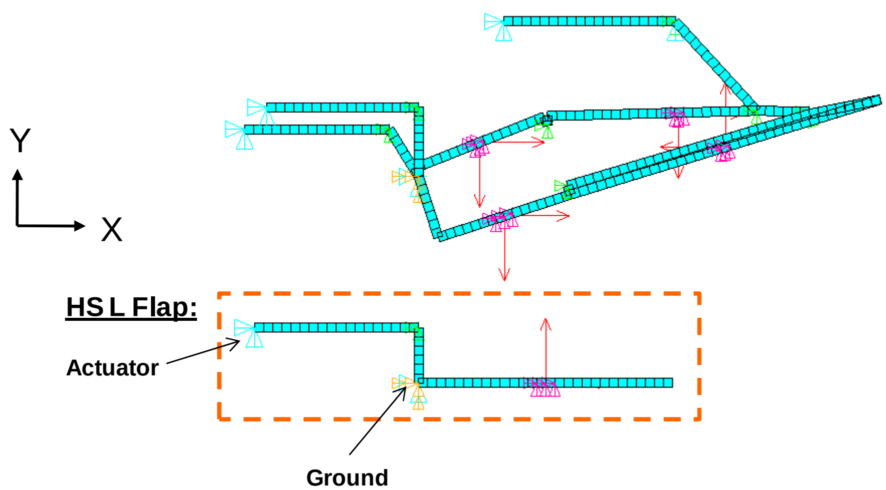
Figure 17: Nozzle Low-Speed Flaps - Force Results Comparison (HiTECC vs. ANSYS FEM)

Results for the high-speed flaps also showed good comparison between HiTECC and ANSYS. Figure 18 and Figure 19 show that all locations considered matched closely with differences below 0.1%.



HS C & D Flap:								
	Actuator					Ground		
Reaction	HiTECC	Ansys	% Diff		Reaction	HiTECC	Ansys	% Diff
RX	-27.8	-27.9	0.36%		RX	17.8	17.9	0.56%
RY	n/a	n/a	n/a		RY	30.6	30.6	0.00%
RZ	n/a	n/a	n/a		RZ	n/a	n/a	n/a

Figure 18: Nozzle High-Speed Flaps - Force Results Comparison (HiTECC vs. ANSYS FEM)



HS L Flap:								
	Actuator					Ground		
Reaction	HiTECC	Ansys	% Diff		Reaction	HiTECC	Ansys	% Diff
RX	152.4	152.3	0.07%		RX	-152.4	-152.3	0.07%
RY	n/a	n/a	n/a		RY	-66.7	-66.7	0.03%
RZ	n/a	n/a	n/a		RZ	n/a	n/a	n/a

Figure 19: Nozzle High-Speed L-Flap - Force Results Comparison (HiTECC vs. ANSYS FEM)

C. Dynamic System Verification

Dynamic verification for the Hydraulics and Kinematics System consisted of observing the proper dynamic trends in forces and position of the LS C-Flap due to changes in nozzle pressure. The system was integrated with the Propulsion System and Thermal Management and Fuel System so that a complete test, including interfaces, could be conducted. The flight Mach numbers and operating modes used during the dynamic verification are shown in Figure 20. This simulated the vehicle accelerating from Mach 2.5 to 4.7, and includes the critical transition period at Mach 3.75 and parts of the turbojet only and DMSJ only operation.

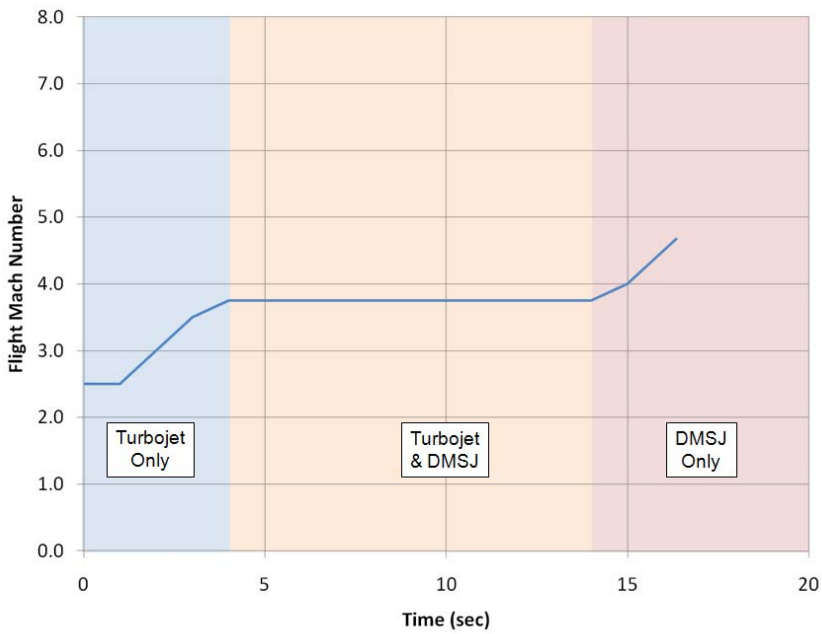


Figure 20. Verification Flight Mach Numbers and Operating Conditions

The low-speed convergent flap controls the throat area of the low-speed nozzle and is used to regulate the turbojet exhaust gas temperature at military up to maximum power and follows an area schedule at part power. The convergent flap position is often monitored with the actuator stroke, which is shown in Figure 21 over the range of flight conditions and operating modes. As the vehicle accelerates during turbojet only operation at maximum power, the flap moves to gradually reduce flow area. From five to six seconds, when the transition from turbojet to DMSJ begins, the flap rotates rapidly to decrease the exhaust area as the afterburner is shutdown. Finally, the flap gradually rotates to its final position at 14 seconds when the turbojet shuts down. The throat area does not fully close since the divergent flap is used to close off the flow path.

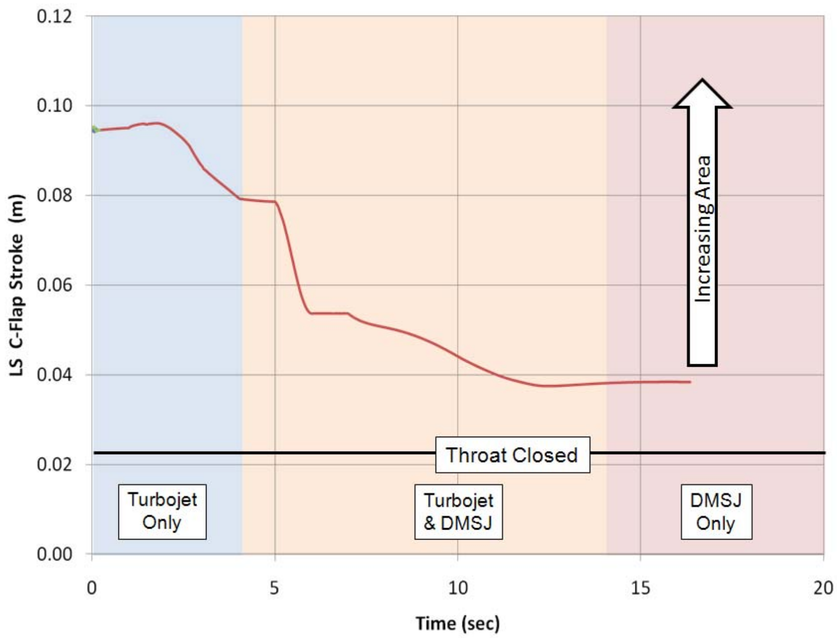


Figure 21. Low-Speed Convergent Flap Position

The low-speed nozzle total pressure and aerodynamic loads on the low-speed convergent flap over the range of flight conditions and operating modes are shown in Figure 22 and Figure 23, respectively. During turbojet only operation, the nozzle pressure rises as the vehicle accelerates, leading to an increase in the magnitude of the aerodynamic loads. At the beginning of transition, the nozzle pressure remains relatively constant as the afterburner shuts down, leading to a fairly constant load. Later in transition, pressure recovery in the inlet improves as the terminating shock moves closer to the inlet throat. This leads to further increases in nozzle pressure and flap load. Finally, the pressure and load begin to drop again as the turbojet completes shut down.

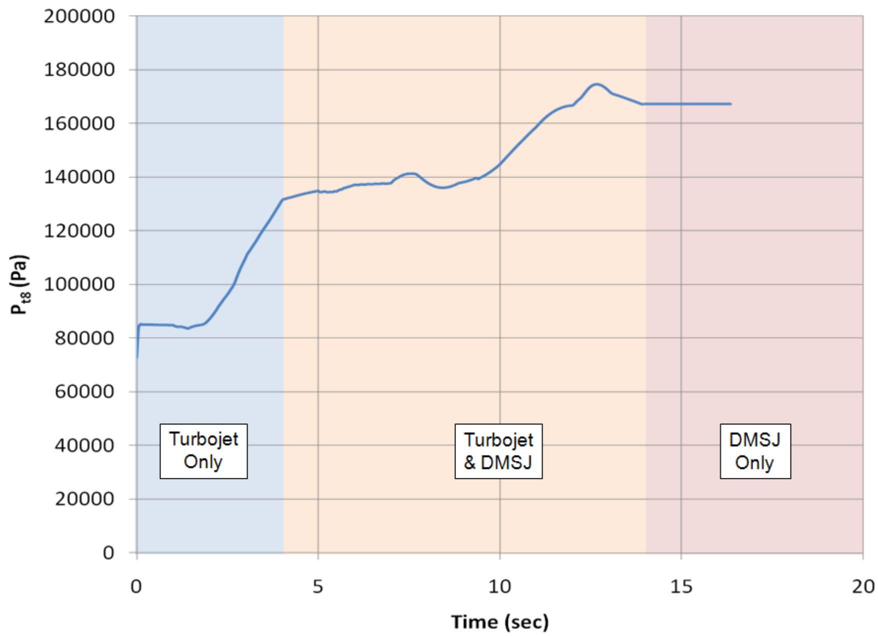


Figure 22. Low-Speed Nozzle Total Pressure

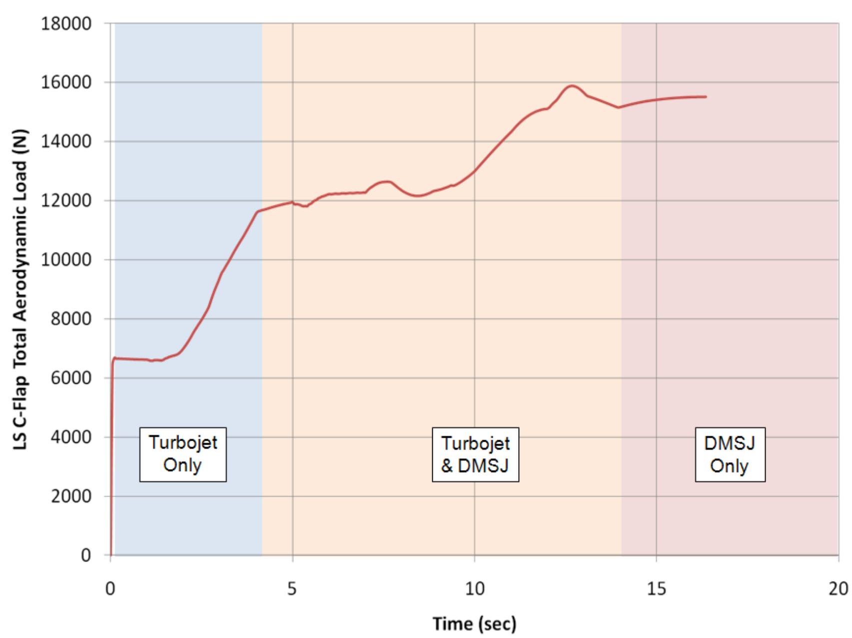


Figure 23. Low-Speed Convergent Flap Aerodynamic Loads

The actuator reaction is shown Figure 24. The reaction load does not mimic that of the aerodynamic load as the distribution of the load between the actuator and ground is affected by the angle of the flap. This is best seen between five and six seconds when the afterburner shuts down. The aerodynamic load is relatively constant, but the actuator load decreases as the flap rotates to reduce the nozzle flow area.

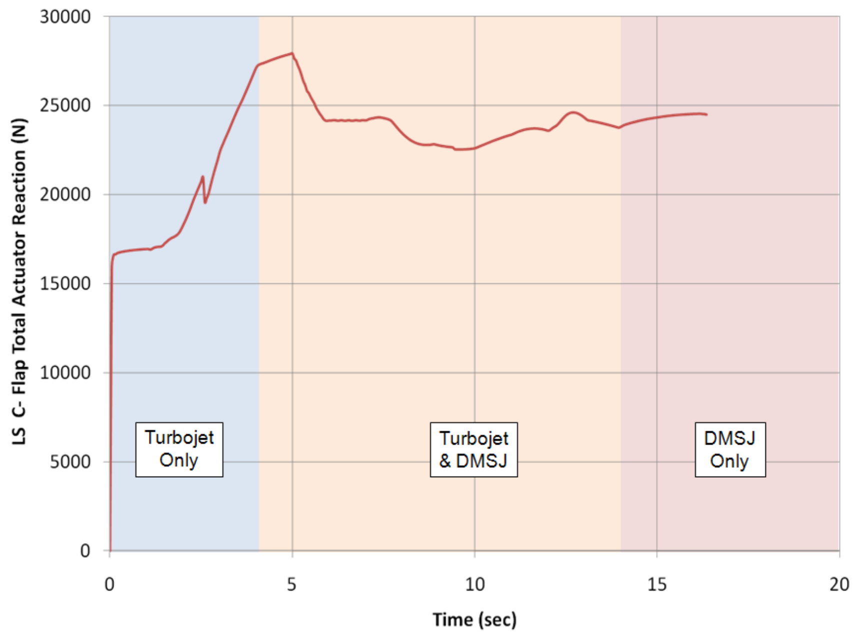


Figure 24. Low-Speed Convergent Flap Actuator Reaction Loads

V. Conclusion

Tools and procedures have been developed for numerical dynamic system modeling of a TBCC propulsion system, including hydraulics and kinematics for both the low-speed and high-speed flow paths. These tools have been incorporated within the High Mach Transient Engine Cycle Code (HiTECC) to computationally simulate a Turbine Based Combined Cycle (TBCC) propulsion system. HiTECC is a dynamic turbine engine model built in Simulink™ to provide real time engine performance predictions during vehicle-wide high-speed transient studies. The model is built on a component level to provide flexibility to model a wide range of engine cycles and to provide internal engine performance data.

The Hydraulic and Kinematic System model is a detailed physical model of the hydraulics and mechanical systems used to actuate variable surfaces for propulsion system control. The model was developed to interface with the Propulsion System to predict real-time pressure loads on the control surfaces. The kinematics model incorporates the aerodynamic loads calculations, along with the dynamic rigid body model, to calculate the reaction loads on the hydraulic actuators. The hydraulic model is a one-dimensional incompressible flow solver that models fluid flow and energy transfer throughout the hydraulic system to predict the hydraulic actuator reaction.

It has been verified that the Hydraulic and Kinematic System developed here can simulate all modes of supersonic operation, including mode transition, for a Turbine-Based Combined Cycle (TBCC) propulsion system. The system has been tested in a simulation of a TBCC vehicle accelerating from Mach 2.5 to 4.7 with mode transition at Mach 3.75. Loads across the kinematic system compare to within 0.2% of finite element predictions for all actuated surfaces. Dynamic response predictions follow trends consistent with expectations for this type of propulsion system.

VI. Future Plans

The development of the TBCC propulsion system dynamic model significantly extends the state-of-the-art for TBCC vehicles by providing a numerical simulation of the propulsion systems for use in future control system development. Future work included in *SPIRITECH*'s TBCC Simulation Model development program includes further integration of the individual systems into the overall HiTECC simulator. The integrated simulator will be used to provide a numerical simulation of the systems, interactions, and transient responses affecting the ability of the propulsion system to transition from turbine-based to ramjet/scramjet-based propulsion.

VII. Acknowledgment

The work documented in this paper is the product of a NASA NRA award under contract NNC08CA55C. The authors would like to thank Dr. Thomas J. Stueber of NASA Glenn Research Center for his guidance and support throughout this effort.

VIII. References

[1] E.J. Gamble , D.A. Haid , and S. D'Alessandro, "Dual-Mode Scramjet Performance Model for TBCC Simulation", AIAA-2009-5298, August 2009.

[2] E.J. Gamble , D.A. Haid , and S. D'Alessandro, "Thermal Management System Model for TBCC Dynamic Simulation", AIAA-2010, July 2010 (submitted for publication).