# Zooming Capabilities of the 1D ESPSS Propulsion Simulation Tool with 3D-CFD Solvers: Implementation and Validation

Kaname Kawatsu and Nobuhiro Yamanishi
*Japan Aerospace Exploration Agency*
JAXA's Engineering Digital Innovation Center (JEDI),
2-1-1 Sengen, Tsukuba, Ibaraki 305-8505, Japan
e-mail: kawatsu.kaname@jaxa.jp

Francesco Di Matteo and Johan Steelant
*ESTEC-ESA*
Aerothermodynamics and Propulsion Analysis Section (TEC-MPA),
Keplerlaan 1, 2201AZ Noordwijk, Netherlands
e-mail: Johan.Steelant@esa.int

For the development of propulsion systems such as the liquid rocket engine, the simulation of the whole system at once using high-fidelity models is computationally expensive. Solely improving individual components is not sufficient as the interaction with the full system is essential. Therefore, one is presently forced to optimize a complete system using a low-order model followed by an isolated high-fidelity optimization for particular components. In the present work, construction and feasibility studies of a new methodology, in which low- and high-fidelity models are operated simultaneously, were conducted. Using a framework for coupling the low-order propulsion system simulation tool EcosimPro/ESPSS with a high-fidelity computational fluid dynamics tool, complete systems can be modeled reliably and accurately within a reasonable amount of time. In this paper, feasibility studies of the coupling of low- and high-fidelity tools are demonstrated.

## 1. INTRODUCTION

EcosimPro/ESPSS is a propulsion system analysis tool perfectly suited in the Concurrent Design Facility (CDF) [1] to evaluate the performance of liquid rocket engine systems within a reasonable time frame. This is a necessary requirement for a concept assessment and system trade-off studies. During a CDF session, a consistent set of design parameters can be defined and exchanged throughout the study, and any changes, which may have an impact on other disciplines, should immediately be identified and collectively assessed. In other words, the performance of a liquid rocket engine system should be evaluated adequately in a shorter amount of time.

EcosimPro [2] platform is an object-oriented tool capable of modeling various kinds of dynamic systems. This tool has a set of libraries with components' models and property functions for the simulation of spacecraft and launcher vehicle propulsion systems, which is called ESPSS [3, 4]. The fidelity of models of ESPSS is based up on empirical formulas, simplified equations and 0D/1D transient equations.

The rocket engine system consists of several components such as turbopumps, valves, injectors, and a thrust chamber. These components are connected to each other with pipes and junctions. This system should behave as a pipe-based system, but some components have complex physics such as combustion, cavitation, and mixtures, which require high-fidelity physical modeling for simulation purpose.

Performing high-fidelity three-dimensional (3D) computational fluid dynamics (CFD) analyses of the entire engine system and provide a comprehensive explanation of the interactions between the components is one possible approach to evaluate complete engine system's behavior [5]. However, because of the prohibitive cost for development and computation, this approach is difficult to be introduced in particular for a concept assessment or system trade-off study within a shorter turn-around time.

EcosimPro is useful for simulating a system's behavior due to its real-time feature and low calculation cost. However, the behavior and performance of some components should be evaluated using a high-fidelity CFD analysis to achieve adequate accuracy.

Therefore, a framework for coupling a system analysis tool, EcosimPro, and a high-fidelity numerical simulation was set-up to combine high accuracy and reduced computational cost. To maintain the real-time feature of EcosimPro/ESPSS, the high-fidelity simulation tool should be adapted to only prime components of a rocket engine system requiring a higher accuracy for system analysis.

The interaction with the CFD analysis allows specific components to be modeled more accurately. Furthermore, the CFD simulation can rely on realistic, transient boundary conditions provided by a low-order representation of the surrounding system. By coupling low- and high-fidelity tools, complete technical systems can be modeled reliably and accurately within a reasonable amount of time.

A feasibility study demonstrating the coupling between EcosimPro and CFD-ACE+ was conducted for a tank and orifice transient system, which is a simplified model of a propellant supply system consisting of a tank, pipe, and valve. In this framework, EcosimPro was used as a master platform for coupling. A high-fidelity simulation with CFD-ACE+ was limited for demonstration purpose only to an orifice.

This work was carried out in collaboration research between ESA/ESTEC and JAXA/JEDI, which is based on Staff Exchange Program between ESA and JAXA. Furthermore, construction and demonstration of the new framework for coupling were supported by ESA/ESTEC.

## 2. COUPLING LOW- AND HIGH-FIDELITY TOOLS

To realize the coupling between low- and high-fidelity tools, several methods can be selected. Improving the accuracy of a whole system analysis may be possible by using a database, which is created from the results of a preliminary high-fidelity simulation that focused on the components of the system. The response surface model (RSM) is a method for creating a database of high-fidelity simulation results. In this technique, the relationship between input parameters and simulation results can be formulated as a function. However, this database may have interpolation errors and the range of the input parameters should be preliminarily predicted to obtain high-fidelity simulation results. Thus, direct coupling was selected in this study. In this direct coupling method, a high-fidelity simulation is conducted during each step of the simulation analysis, which uses a low-fidelity tool.

### 2-1. EcosimPro/ESPSS

EcosimPro is an object-oriented tool capable of modeling various kinds of dynamic systems. EcosimPro has a set of libraries with component models and property functions for the simulation of spacecraft and launcher vehicle propulsion systems, which is called ESPSS. In this framework for coupling low- and high-fidelity tools, EcosimPro/ESPSS assumes the central role as a main platform, which controls the procedure of the simulation, using C++ library functions. The C++ library function enables the platform to make an interface with the 3D-CFD tools to exchange information between the low- and high-fidelity tools, which realizes the coupling.

### 2-2. 3D-CFD tool

CFD-ACE+, which was originally developed by the CFD Research Corporation (CFDRC) and is currently owned and distributed by ESI [6], was selected as a 3D-CFD simulation tool for a feasibility study for the coupling of low- and high-fidelity tools. CFD-ACE+ is a multi-physics and multi-disciplinary simulation tool, which has a flexible interface with an outer program using a journaling function in the Python programming language. This journaling function enables control of the simulation process and setup for the simulation conditions by an outer outer program.

### 2-3. Framework for the coupling

Figure 1 shows an overview of the framework for the coupling between the low- and high-fidelity tools. In this framework, an I/O module described in C++ exchanged data between EcosimPro and CFD-ACE+, and controlled the execution of CFD-ACE+ during the complete system analysis procedure by using a journaling function of CFD-ACE+, which controlling the setup for boundary conditions.

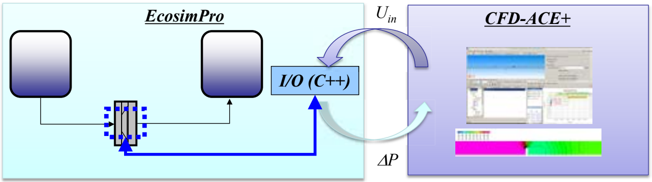


**Figure 1. Overview of framework**

The whole procedure of the coupling framework was divided into five steps as shown in Fig. 2. These procedures were conducted by using a C++ library based on EcosimPro platform.

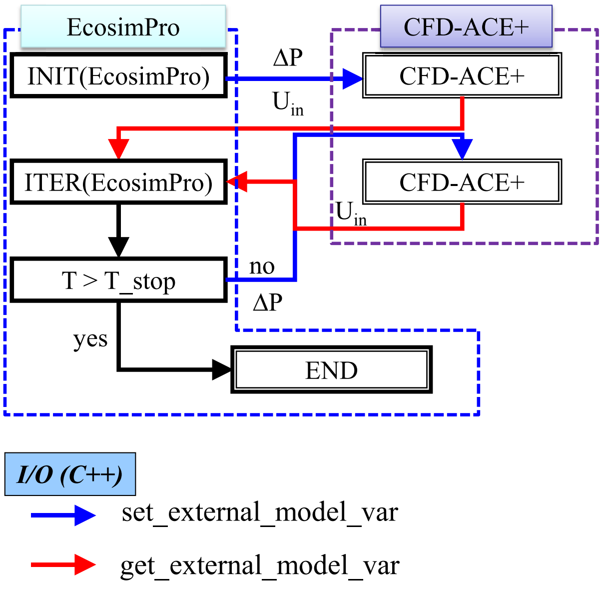


**Figure 2. Flowchart of framework**

Step1. Initialization and setup of CFD-ACE+

In this framework, the initialization and setup for the 3D-CFD simulation using CFD-ACE+ was conducted by the journaling function. In this procedure, a journal file, which contained all of the required information for the CFD-ACE+ simulation, was created in the Python programming language.

Step2. Invoke and execute CFD-ACE+

One of the C++ libraries on the EcosimPro platform, which was named "set_external_model_var," controlled an execution command to invoke CFD-ACE+. High-fidelity simulation using CFD-ACE+ is carried out with setup, which was created in the Python programming language including boundary condition information determined by EcosimPro platform. This high-fidelity simulation will evaluate inlet velocity on orifice part in the case of feasibility study, which was carried out in this work.

Step3. Check a result of CFD-ACE+

To determine the next time step value for an EcosimPro side calculation, this framework monitored the CFD-ACE+'s output file, which recorded the inlet velocity at each calculation step using a C++ library, which was named "get_external_model_var." If the CFD calculation converged, the inlet velocity value, "Uin," was transferred to EcosimPro.

Step4. Calculation of Ecosim with new "Uin" value

EcosimPro performed a next time step calculation using the "Uin" value, which was determined by the high-fidelity CFD simulation using CFD-ACE+. In this part, system's transient behavior is evaluated by EcosimPro.

Step5. Update setup for CFD-ACE+

The inlet pressure drop (ΔP) calculated by result of transient system analysis using EcosimPro was transferred to CFD-ACE+ to set the new boundary condition of inlet pressure using the C++ library "set_external_model_var". This setup is also conducted by the journaling function of CFD-ACE+.

These five steps were controlled automatically by EcosimPro during the whole analysis. Thus, CFD-ACE+ is treated as a component model of EcosimPro.

## 3. Feasibility Study of Framework

In this section, detail and results of feasibility study for the new framework are discussed. This feasibility study is carried out to demonstrate a feature of the framework to achieve reliable and accurate numerical modeling of a system within a reasonable amount of time.

### 3-1. Target

A feasibility study demonstrating the coupling between EcosimPro and CFD-ACE+ was conducted for a tank and orifice transient system representing a simplified model of a propellant supply system, which consists of a tank, pipe, and valve, as shown in Fig. 3.

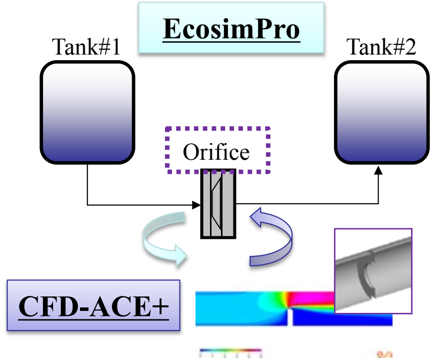


**Figure 3. Schematic of tank and orifice system**

In this study, high-fidelity simulations using a 3D-CFD tool were employed only for the orifice component, which was assumed to be a valve of a propellant supply system. The other parts of the system, i.e. tank and pipes, were modeled by EcosimPro.

### 3-2. Numerical setup for system analysis

Figure 4 shows the numerical setup of the tank and orifice system for EcosimPro. This model consists of two tanks and one pipe. The pressure drop ($\Delta P$) in the orifice pipe depends on the inlet velocity of fluid ($U_{in}$). The relationship between $\Delta P$ and $U_{in}$ was evaluated using a 3D-CFD tool, CFD-ACE+, in this coupling framework.

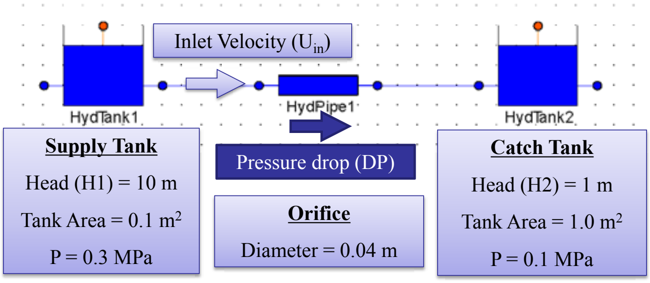


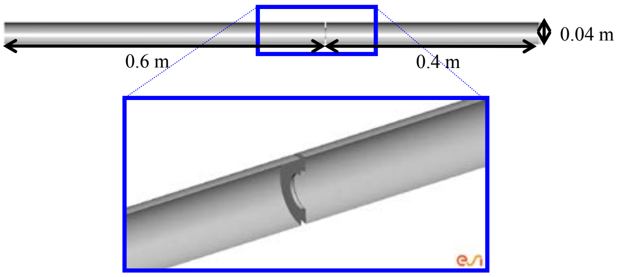
**Figure 4. Numerical setup for EcosimPro**

In the procedure of the EcosimPro simulation, the pressure difference between the inlet and outlet of the orifice (HydPipe1) was calculated using the heads of the supply tank and catch tank. The inlet velocity, which was evaluated by CFD-ACE+, determines a flow rate passing through the orifice pipe. The tank heads at the next time step were calculated using this flow rate.

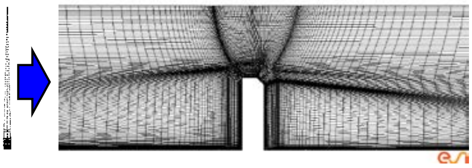
### 3-3. Numerical model and condition for 3D-CFD

A 3D-CFD simulation was adapted to an orifice component, which is a simplified model for a valve in propellant supply system. When fluid flowed through the orifice, a pressure drop occurred as a result of the resistance to flow. This pressure drop was evaluated by CFD-ACE+.

Figure 5 shows the computational region and model of the orifice component. This model includes the inlet and outlet pipe of the orifice, which interface with the supply tank and catch tank in the EcosimPro model.



Configuration of orifice duct



Topology of grid : All Hexahedra

**Figure 5. Computational region and model**

Table 1 shows the CFD method and condition for the orifice component. The steady state simulation was performed with the inlet pressure, which was determined from EcosimPro's result, to evaluate the inlet velocity of the orifice component. The velocity of the fluid was transferred to the EcosimPro model after the simulation convergence was confirmed by the framework's monitoring function. The monitoring function checked for a difference of the inlet velocity value between the previous and present iteration step.

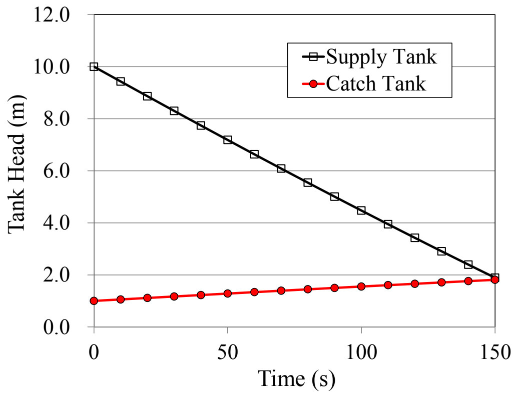
**Table 1. CFD method and condition**

| CFD Method | |
|---|---|
| Solver | CFD-ACE+ Ver.2010 |
| Module | Flow, Turbulence (k-e Standard, Standard Wall) |
| Spatial Differencing | Velocity 2nd Order, Turbulence 2nd Order |
| Transient Conditions | Steady |
| **CFD Condition** | |
| Fluid | Water |
| Inlet BC | Static pressure = From EcosimPro,<br>Static temperature = 293 K |
| Outlet BC | Static pressure = 0.100 MPa |
| Wall BC | No-slip |

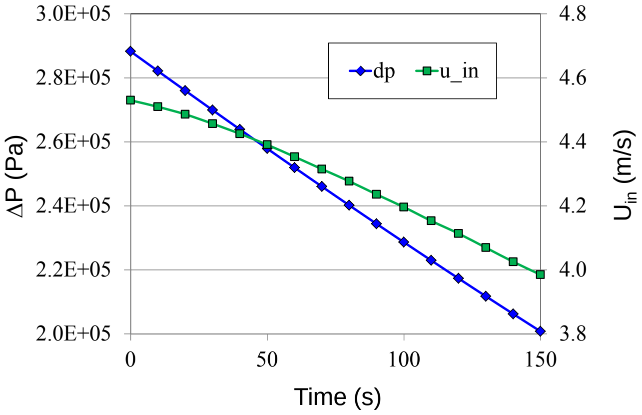
## 3-4. Results and discussion

Figure 6 shows the time history of an EcosimPro output. The tank and orifice system's transient behavior, which was observed as variations of the head, was simulated using EcosimPro. Initially, the head of the supply tank was higher than the head of the catch tank. This difference in head between the supply tank and catch tank decreased with the flow of fluid through the orifice. The flow rate of the fluid was determined by the flow velocity. This velocity was evaluated by a high-fidelity simulation using CFD-ACE+ with a numerical setup by the EcosimPro C++ library at each time step of the EcosimPro analysis. The inlet boundary condition of pressure for the CFD-ACE+ analysis was updated using EcosimPro's result to evaluate the inlet velocity at each time step of the unsteady system analysis, which was performed by EcosimPro.

Figure 7 shows the time history of the pressure and velocity distributions, which were evaluated by a high-fidelity simulation using CFD-ACE+. At each time step, the inlet and outlet pressure were fixed as boundary conditions. The distribution of the velocity in the orifice component was adequately predicted from a physical point of view by the high-fidelity simulation. This adequate result of the inlet velocity value was applied to the next time step of the EcosimPro system analysis to evaluate the change in the tank head value. These procedures of coupling between the low- and high-fidelity tools were achieved by the new framework constructed in this study.

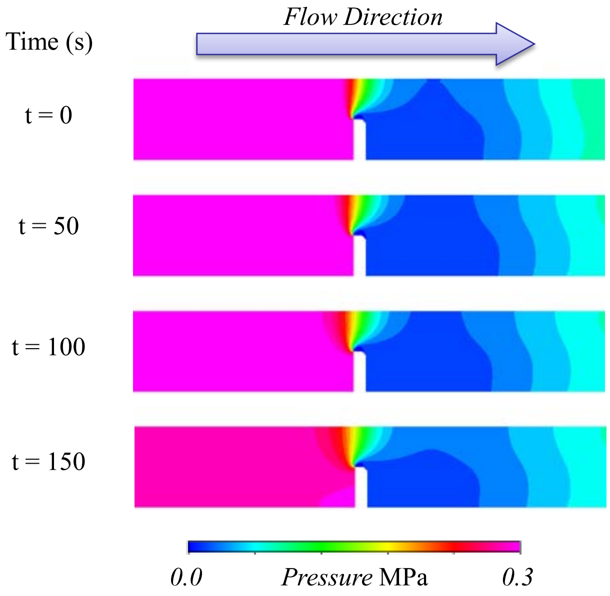


(a) Tank Head

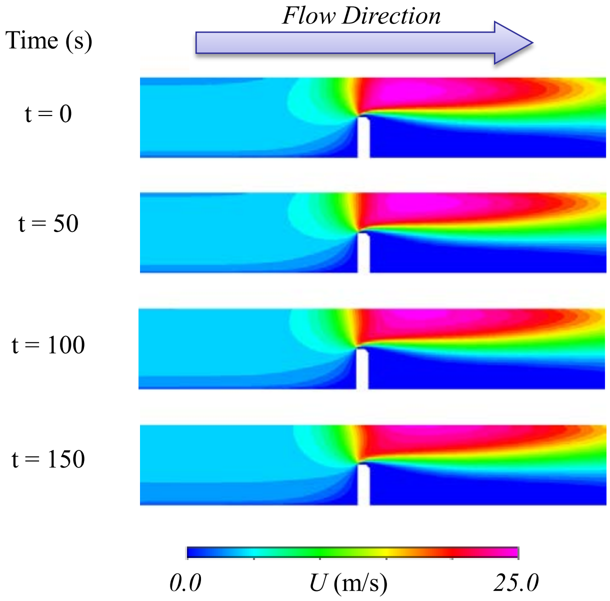


(b) Pressure Drop and Inlet Velocity

**Figure 6. Time history of EcosimPro output**



(a) Pressure Distributions



(b) Velocity Distributions

**Figure 7. Time history of CFD-ACE+ output**

## 4. OUTLOOK AND FUTURE APPROACH

The numerical procedure performed and described in this paper stimulates for further improvements and developments to increase the reliability and the capabilities of the coupling framework. Next plans envisage covering more complex simulations elaborating the interaction of CFD with the EcosimPro platform:

- Coupling between ESPSS and the CFD Tau code for launcher propulsion in general. CFD Tau code is mainly a CFD tool for external hypersonic flow simulation and particular combustion processes such as for scramjets [7]. Application of the coupling could be base-flow LRE nozzle interaction along the trajectory or intakes/nozzles for liquid flying back boosters where ESPSS is used for the turbo machinery, etc.
- Coupling between EcosimPro/ESPSS and CFD-ACE+ code to evaluate multi-disciplinary aspects such as conjugate heat transfer between rocket engine combustion chambers and cooling channels.

## 5. SUMMARY

In this study, a new framework to achieve reliable and accurate numerical modeling of a system within a reasonable amount of time was constructed. In this framework, the low-order propulsion system simulation tool EcosimPro/ESPSS was coupled with the high-fidelity CFD tool. EcosimPro functioned as a master platform of the framework for the coupling. The high-fidelity CFD tool simulated only a component part using a numerical setup, which was defined by EcosimPro’s result. In addition, the CFD result was used as the condition for the next time step calculation of EcosimPro's transient simulation. The information exchange between EcosimPro and the CFD tool was presented by using a new C++ library of EcosimPro.

A feasibility study demonstrating the framework for coupling EcosimPro and CFD-ACE+ was conducted for a tank and orifice system. This system is a simplified model of a propellant supply system, which is constructed of a tank, pipe, and valve. The transient simulation using the framework demonstrated the feasibility of the framework for coupling the low- and high-fidelity tools.

## ACKNOWLEDGEMENTS

The construction and feasibility study of the coupling framework in this study were conducted within the staff exchange program between ESA/ESTEC and JAXA/JEDI, which was supported by Dr. Johan Steelant (TEC-MPA/ESA-ESTEC) and his colleagues. The author greatly appreciates their contributions and support.

## NOMENCLATURE

$P$: pressure

$T$: temperature

$U$: velocity

(Greek letters)

$\Delta$: difference

(Subscripts)

$in$: inlet

## REFERENCES

[1] ESA/ESTEC web site: http://www.esa.int/esaMI/CDF/, 2011

[2] EcosimPro. Modeling and Simulation Software. Installation & Getting Started Guide. Version 4.8.

Madrid, EA International, 2010.

[3] Empresarios Agrupados, “ESPSS EcosimPro Libraries User Manual”, 1.4.1 edition, 2009.

[4] M. De Rosa, J. Steelant, and J. Moral, “ESPSS: European Space Propulsion System Simulation”, In Space Conference, May 2008.

[5] N. Tani, et al, “An End-to-End High Fidelity Numerical Simulation of the LE-X Engine - Engine Performance Evaluation and Risk Mitigation Study - ”, AIAA-2011-5931, 47th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, San Diego, California, July 31-August 3, 2011.

[6] CFD-ACE+, Version 2010. User Guide. Huntsville, AL, ESI-CFD Inc., 2010.

[7] Homepage of DLR TAU Code, DLR Institute of Aerodynamics and Flow Technology. : http://tau.dlr.de/, 2012.