Coupling 1D and 3D CFD: Myth or Reality

Vincent A. Soumoy
(Fluids & Co, Canada);

Member of the NAFEMS CFD Working Group

Abstract

For years Computational Fluid Dynamics (CFD) has been widely used across many branches of engineering including the Aerospace, Automotive, Biomedical Chemical, Marine, Oil, Gas, Petrochemical and Power Generation industries.

One dimensional (1D) CFD allows engineers to understand how flow rates and pressures may change within a network flow system of interconnected components. In contrast, three dimensional (3D) CFD allows design engineer to understand how detailed flows interact with all manner of complex geometry. Of course, 1D calculations are much faster, only a few minutes, when 3D CFD may take hours or days.

Co-simulation of a 1D with a 3D model will potentially offer the best of both worlds. It enables the sharing of boundary data between 1D and 3D models in a single or multi-domain system to facilitate the simulation of the overall fluid system coupled with more detailed CFD simulation of 3D flows within a critical part of the network.

In March 2010, NAFEMS organised in Gaydon (UK) a seminar to present the state-of-the-art of the links. Most speakers (ANSYS, CD-adapco, LMS, Flowmaster, AVL, SCAI) presented the various possible links between 1D (Flowmaster, Amesim, GT Power…) and 3D tools (Fluent, CFX, STAR-CCM+, FIRE…). Some industrials also showed examples, but most of them only referred to simplistic applications, only to demonstrate the capabilities of the link.

We are now nearly 10 years later, tools and editors have changed, or merged.

Is 1D and 3D co-simulation now a reality?

We saw in 2010, that the 1D codes were driven by the engine approaches, in automotive and aerospace sectors, but most of the other industries were interested by co-simulation. 1D-3D links were already 10 years old (Fluentlink for Flowmaster arrived in 2001), but a lot of work was still to be done.

The CFD Working Group (namely David Kelsall and the author) were involved in the 2010 seminar. All the speakers have been contacted and agree to show that 1D-3D is now a reality.

1. Introduction – the concept

The initial typical concept is quite simple, to allow 3D CFD simulation to collaborate with 1D CFD or systems simulation. The 3D option is local and detailed but slow, the system consideration is fast but general and do not offer detailed information on typical components.

We had to consider two kinds of industrial need: the 1D user who is looking for a local detail or the 3D user who would like the more real as possible boundary conditions.

3D analysis is being used where the nature of the flow and the understanding required makes 3D analysis the appropriate tool; 1D simulation is applied to examine the fluid flow conditions of the remaining system, which can be captured by a 1D calculation with in-built sub-models for specific components as required. Boundary conditions and results are then passed around the complete system, allowing for a more complete and faster analysis. A link is in charge to handle communication of variables (and results) between the models. Most of the software vendors have to enable users to bi-directionally link their 3D CFD models (often via a simple to use and intuitive user interface) to a 1D fluid flow system network. This 1D network then analyses for pressures, flows and temperatures across the whole system, reporting boundary conditions (Steady Sate or Transient) directly back into the CFD model.

The 1D system can also allow thermal interactions or mechanical positions from other systems and incorporate those effects when determining the component CFD boundary conditions.

To ensure proper functioning of the coupling, the interface must not be located in a region in which flow separation or recirculation is expected and must be a planar face orthogonal to the expected main flow direction.

This process will also allow you to investigate the influence of a complex three-dimensional component on 1D fluid flow and system performance, to provide detailed information about 3D flow fields in the domain of interest; considering the influence of all relevant sub-systems modelled one-dimensionally and to obtain an immediate response of the sub-systems on design changes.

We will review the first attempts of the 1D & 3D software vendors to reach these capabilities, the situation about 10 years ago, at the initial NAFEMS seminar on this subject, and the actual offers. We will also mention some external ways used for the same target.

2. The early beginning

In 2001, Flowmaster (the most common 1D fluid code at this time) and STAR-CD (from CD-adapco) distributed a direct link (called STARLink) initially for the German automotive industry. It was followed by a FluentLink for Fluent (from ANSYS).

Subsequently similar links were developed between Fluent, STAR-CD and AMESIM (from LMS). In the meantime, STAR-CD was also linked with GT Power (from Gamma Technologies) and WAVE (from RICARDO).

AVL developed their own interface between the 3D code FIRE and the 1D codes BOOST (from AVL), GT Power and WAVE. In the automotive engines application, the use of coupled 1D/3D simulation was highly motivated by the investigation of the impact of design modifications on the engine performances, namely for the EGR (Valve used to control pollutants in exhaust systems) supply location or the lambda probe positioning.

GT-Power, coupled by Gamma Technologies with STAR-CD, was used by the University of Stuttgart in a wide range of application from tasks such as the design of switch-over intake systems, to detailed flow analysis.

3. The situation at the Gaydon seminar in 2010

In March 2010, NAFEMS organised in Gaydon a seminar where the various options in the 1D-3D domain were presented.

Dominik Sholz from ANSYS Germany gave detailed information on the different links available and showed some examples.

He reminded us that the Fluentlink was developed between Flowmaster and Fluent for vehicle thermal management and started the series. He mentioned that Fluent was linked with GT Power, Wave and Flowmaster. CFX is linked with GT Power and Amesim. In some cases, the link required custom code.

He showed an example, of Fluent and Flowmaster describing a water jacket cooling. With the co-simulation, pressure drop through the cooling jacket increased by 12%, flow rate through radiator increased by 13%, flow distribution through the turbo, bypass, heater core were influenced and pump operated at a different operating point.

As a second step, the link between Fluent and GT Power or Wave was described. CFD components were defined in the 1D code and boundary macros were automatically created to hook to the CFD boundaries and to exchange variables like mass flow rate, pressure, temperature, and species mass fractions.

A vehicle thermal management model was presented, running simultaneously Fluent, GTPower and Flowmaster. Results are compared with experiments and the difference in temperature remains lower than 2%.

The next part of the presentation concerned co-simulation between CFX and Amesim. Examples with ball valve, cryogenic pipe and tracer transport with 3D effects in TT junction were described.

To finish an intake manifold (EGR) was described between CFX and GT Power.

David Kelsall, from Flowmaster, developed a co-simulation for large aircraft environmental system between Flowmaster and Fluent or STAR-CCM+.

Using the best of both worlds was the main idea of this model. The system model is fast to set up and very quick to analyse. Moreover, the 1D air distribution system modelling will allow analysing large aircraft air distribution, to evaluate duct re-routing scenarios, to study the mixing of fresh and re-circulated air and to conduct “what-if” scenarios on duct sizing.

It also enabled the understanding of system interactions, helped to meet passenger flow rate requirements and guaranteed comfortable temperatures and pressures.

The 3D analysis was a better technique for modelling larger „open" volumes with complex flows and for giving better appreciation of factors affecting passenger comfort.

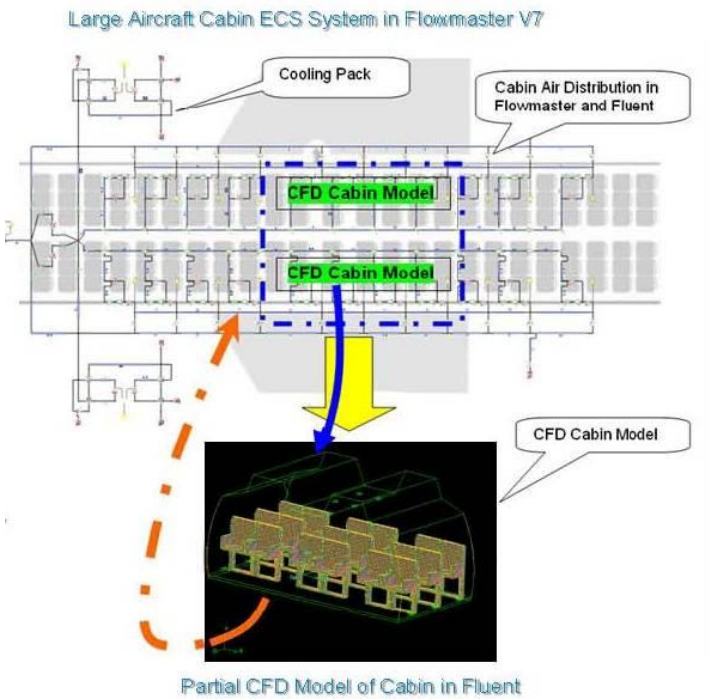


Figure 1: The Fluent-Flowmaster ECS model.

The results obtained were challenging and stimulating and showed that changes in 1D model would directly affect 3D model.

This first approach on industrial case presumed that realistic simulations were possible and gave scope for further development and optimization, but David Kelsall remembered that validation is important.

STARLink (for STAR-CD), developed in 2001 by CD-adapco and Flowmaster was the first 1D/3D link, initially for the German automotive industry.

Sreenadh Jonnavithula from CD-adapco was presenting the new link between STAR-CCM+ and Olga, GT-Power and Wave.

The way CD-adapco has developed those new links corresponded to the ideal solution for users. They did not need intermediate tools between 1D and 3D codes and did not have to worry about the coupling itself.

The goal was to provide a consistent interface for coupling multiple 1D codes, assuming that 1D problem is set up by an expert user of 1D code.

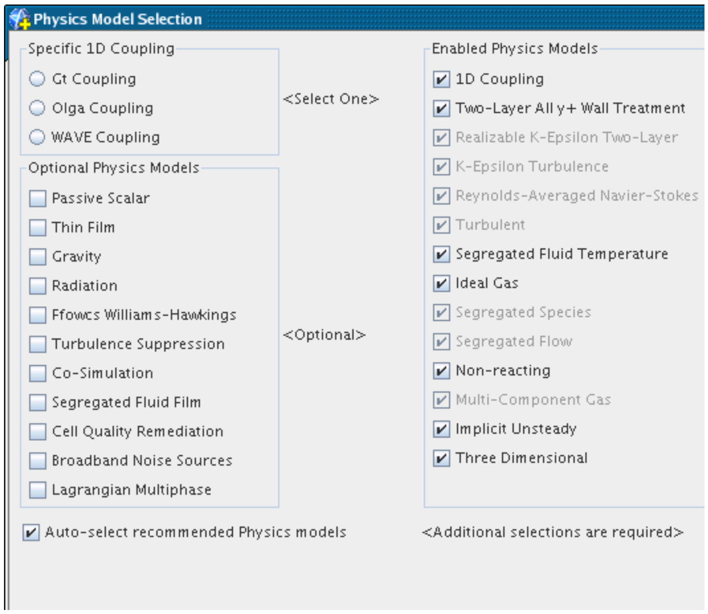


Figure 2: The STAR-CCM+ interface with the 1D Coupling.

CD-adapco provided tools in STAR-CCM+ to : choose the 1D code to use, indicate boundaries to be coupled to 1D code (appropriate variables coupled depending on 1D code), associate solution variables which might have different names in the codes, send control information to 1D code (file locations e.g.), hide the complexity of flow & time step control, boundary condition averaging or distributing etc., (depending on 1D code) from user.

Examples shown coupling STAR-CCM+ with Olga, GT-Power and wave suggest a lot of new application in future for example with lagrangian particle tracking.

4. The MpCCI option

Around 2010, MpCCI (Mesh-based parallel Code Coupling Interface) has been developed and distributed by Fraunhofer-Institute for Algorithms and Scientific Computing (Fraunhofer SCAI) to support Abaqus, ANSYS, Fine/HEXA, Fine/TURBO, Flowmaster, Fluent, Flux, ICEPAK, MSC.Marc, Permas, STAR-CD and RadTherm amongst others.

MpCCI overcomes the challenges inherent in co-simulation - complex hardware requirements and challenging software engineering requirements - by using adapters (developed by each software vendors) to establish a direct connection between the MpCCI Coupling Server and the 1D or 3D CFD code.

This new interface could facilitate interactions between different software tools as it remains partially vendor independent but was probably less convenient than the former direct links. Indeed, the initial links here were created for specific 1D and 3D codes, so they are fully dedicated. The graphical user interface was user-friendly and often directly included in the 1D user interface. The only problem was that each 1D or 3D software was live; the version was regularly changing, and the link was generally dependent on both 1D and 3D code version.

Using MpCCI avoided the main part of this dependence but as it was a general tool, the user interface was less easy to use than the direct ones.

Pascal Bayrasy showed at Gaydon how realistic simulations be. He first presented the various advantage of this vendor independent tool, allowing multi-software and multi-systems models. Some typical examples for fluid structure interaction on a turbo charger fan, for radiation coupling with RadTherm and for full vehicle thermal management were presented.

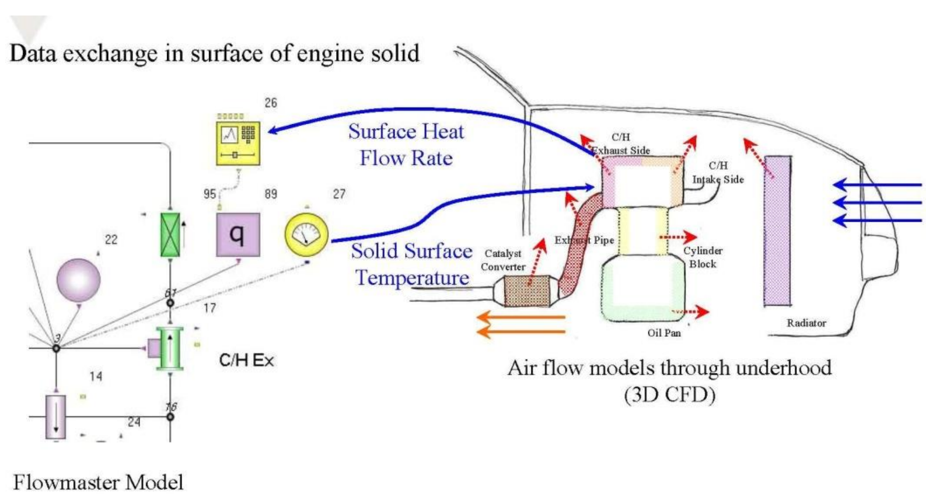


Figure 3: A coupling example presented at Gaydon by with MpCCI.

The implementation was easy to set up but we could presume that for highly complex systems some convergence problems could occur.

5. Another option, the external workflow process.

Today, with the same kind of method that the one used by MpCCI, some tools allow to manage all the logical steps of your engineering process with a single automated workflow.

Complex engineering problems often include the use of a myriad of in-house and third-party CAD, CAE and general use software, including 1D and 3D CFD, resulting in a disconnected, difficult to manage process.

Integration and process automation tools, like modeFrontier, for example, will streamline and automate the engineering process within an integrated workflow of various software to increase the overall efficiency, save time and reduce operational costs.

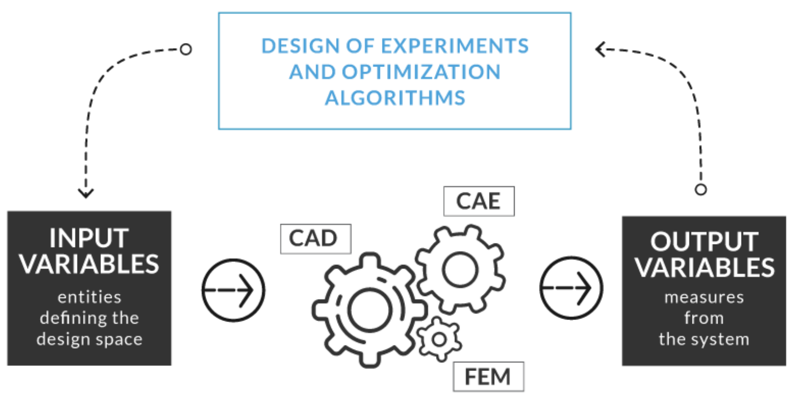


Figure 4: How to integrate and drive multiple CAE tools

They will streamline the execution of complex simulation processes and effectively manage system-level optimization.

Complex engineering problems include the necessity to take into account multiple disciplines, consider high number of interrelated variables, and run multiple third-party simulation software.

The automated, workflow-based environment for multidisciplinary optimization enables the efficiently execution of complex integration chains, the breaking of large engineering problems into modules, and the scheduling of nested optimizations from within the main workflow to improve the overall efficiency of the design process.

The technology offers a variety of direct integration nodes to couple with the most popular engineering solvers.

Direct nodes seamlessly connect with the simulation models, extract relevant parameters and allow the selection of the process input and output in few clicks, through a simple guided process. Constraints and objectives just have to be added to run the process and exploit the benefits of process automation.

Mainly most of the 1D and 3D CFD software vendors are involved ANSYS (Fluent, CFX), AVL(Ast), GAMMA (GT-Suite), SIEMENS (STAR-CCM+, FloEFD, FloMaster, Amesim), EXA (Powerflow)…

6. The actual situation, two main examples

Today, two 1D leaders are offering 1D-3D co-simulation: FloMaster (the former Flowmaster) and Flownex.

The approach used by Simcenter FloMaster and Simcenter FloEFD is different than past approached as it does not require a middleware program to manage the passing of data between programs and sequencing time steps. On the contrary, the tools share a common solver matrix and both models are solved simultaneously. This provides simulation efficiencies throughout the process by reducing the time to set up the co-simulation, reduced data handling and simpler post processing.

For the first time in the industry, Mentor Graphics was providing a tightly coupled general-purpose 1D-3D CFD simulation software combination. With this combination, engineers can characterize the more complex elements of the system with full 3D and easily insert those models into the 1D system level model for simulation.

FloEFD Concurrent CFD can reduce simulation time by as much as 65–75% in comparison to traditional CFD tools. Concurrent CFD enables design engineers to optimize product performance and reliability while reducing physical prototyping and development costs without time or material penalties.

And because FloMaster offers a fast and reliable solution to 1D flow problems, designers can expect unrivalled savings in both time and process compression.

Mentor Graphics 1D-3D solution provides an opportunity to increase digital prototyping information earlier in the product development process. This kind of analysis performed upfront accelerates design and development cycles by orders of magnitude and optimizes product design workflows. It increases engineer and designer productivity and minimizes design risk and re-spins.

This 1D-3D combination provides the best of both worlds, thus reducing time to get essential component information to system engineers and improving system-level accuracy with highly complex geometries, thus reducing the design cycle time.

To combine the best of both worlds, Mentor offers three new capabilities that will be presented in a dedicated paper (Croegaert and al).

Simulation Based Characterization is a methodology that enables complex multi arm geometries to be characterized using the power of 3D Computational Fluid Dynamics. FloEFD is used to perform the 3D characterization in a mechanical cad environment using a (computational) design of experiments and response approach to extract hydrodynamic and thermal response surface models that are then imported into FloMaster and represented with an 'N-Arm' component.

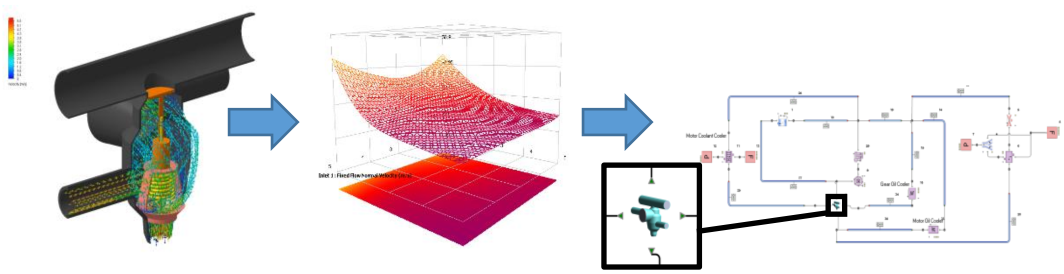
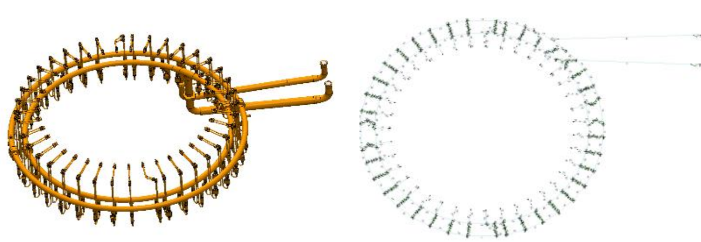


Figure 5: The Simulation Based Characterization from Mentor Graphics

Cad to FloMaster (CAD2FM) is a fully automated function to convert MCAD piping geometry into an equivalent FloMaster sub-system, bypassing data re-entry errors and compressing the time required for such a conversion from hours to minutes compared to manual 'measure and re-enter' approaches. This entails the use of FloEFD to prepare the geometry for export, perform the conversion using a simple 'coarse/fine' slider bar to control the level of abstraction, then export of a



.cad2fm file containing the FloMaster sub-system. The .cad2fm file is then imported into FloMaster and saved to the sub-system catalogue for subsequent reuse.

Figure 6: The CAD2FM tool from Mentor Graphics

And last tool, OneSim is a tightly coupled co-simulation workflow that enables a 3D FloEFD model to be considered as part of a FloMaster system simulation. First, one or more FloEFD boundary conditions are linked to nodes within a FloMaster network. Then a simulation instigated from within FloMaster will solve both the FloEFD model and the FloMaster network concurrently, until steady state or transient convergence is achieved. Flow rates, pressures and fluid temperatures will be communicated through the linked boundary conditions nodes throughout the solution process. An implicit coupling technology is used to enable solutions to be achieved more readily compared to more classic co-simulation approaches that can be prone to instabilities, often requiring middleware to govern the solver communication and can be notoriously tricky to set-up.

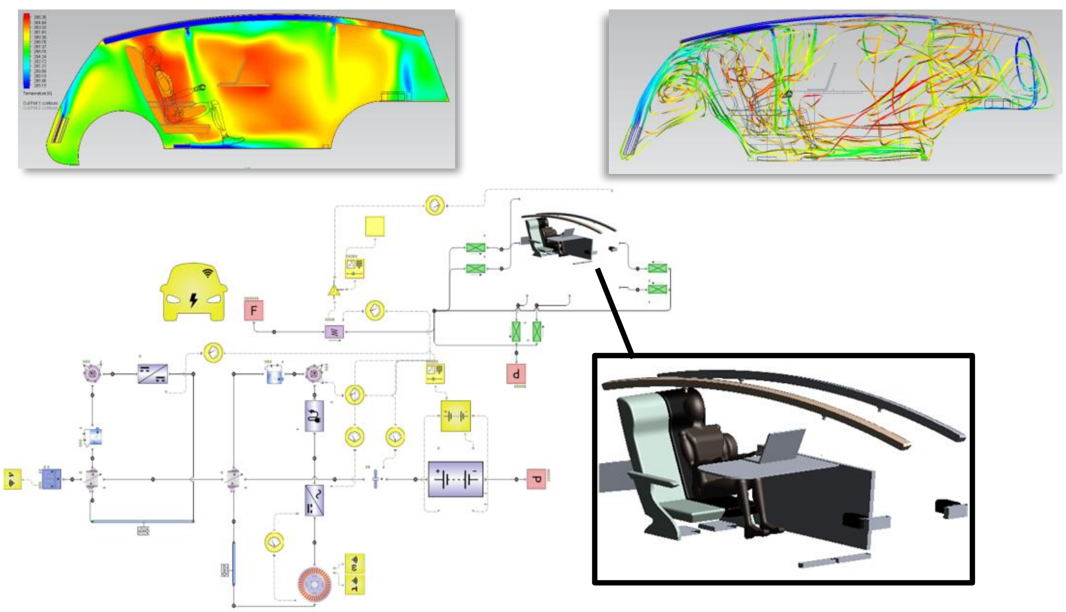


Figure 7: The Onesim tool from Mentor Graphics

It was easier to combine those technologies in the SimCenter as now Mentor Graphics (presently a Siemens Business) was owner of the 1D and 3D software.

On the other side, Flownex made the choice to collaborate with ANSYS and decided to integrate their 1D system tool to the workbench environment.

Flownex is linked to both ANSYS CFD and FEA codes for more localized results where required.

In particular Flownex can be coupled with ANSYS Fluent to provide an interactive communication between a CFD 1D system and CFD 3D analyses. This allows the inclusion of the effect 3D complex geometries in terms of pressure losses, multiphase flows and non-homogenous heat transfer.

Flownex can also be coupled with the FEA code ANSYS Mechanical: thermal and pressure stress analyses can be performed starting from the temperature and pressure calculated by Flownex. Co-simulations between 1D network and FEA 3D code can be used to model conjugate heat transfer with 3D geometries: this approach keep the accuracy of temperature distribution in the solid domain and reduce the computational effort on the fluid side.

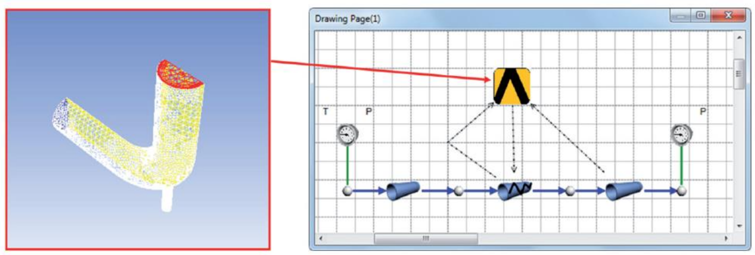


Figure 8: 1D and 3D flows are solved together, and information is exchanged at the boundaries. The entire simulation is controlled by Flownex interface

One application of the coupling with ANSYS Fluent can be illustrated in the simulation of the HVAC system in a server room. In this case Flownex is used to simulate the fan, heat exchanger and ducting to the server room while ANSYS Fluent is used to simulate the heating of the air by the electronics and the air flow in the room.

The interface points are chosen at the vents that supply air to the room, where the temperature and flow results from Flownex are transferred to Fluent and the backpressure results from ANSYS Fluent are returned back to Flownex. In this simulation, control elements are added to automatically adjust the fan speed and the vent openings in order to study the transient scenario of the system.

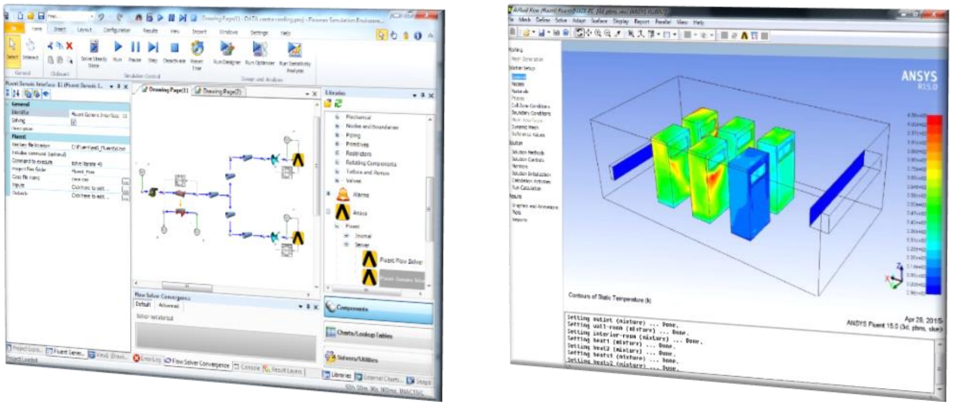


Figure 9: Flownex and ANSYS Fluent link: HVAC system in a server room

Flownex SE also adds value to structural simulations and can be used to transfer 1D flow results to a FEA simulation package such as ANSYS Mechanical.

The ANSYS Mechanical coupling finds application in many industries. An example of the ANSYS Mechanical coupling in the power generation industry is illustrated below.

Figure 10: An example of typical boiler tube failure locations

It shows a model of a boiler where the flows are calculated using Flownex and the thermal stresses are calculated using ANSYS Mechanical. Using this analysis technique, start-up conditions can be simulated in a transient environment and boiler design can be modified to ensure the stresses are all within allowable limits before construction begins.

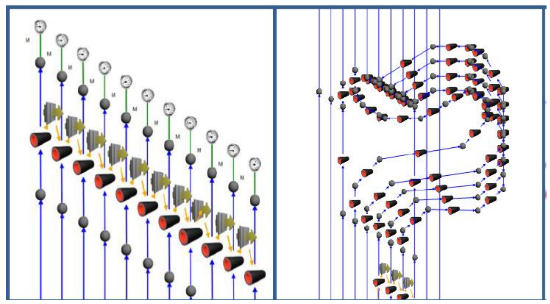


Figure 11: Zoom view of some of the custom compound components illustrating the network that was modelled in Flownex

Flownex can also calculate pressure forces on elbows and pipes: these can be used as an input to mechanical design for the calculation of pressure stress analyses and frequency analyses based on pressure signal (using Fast Fourier Transform).

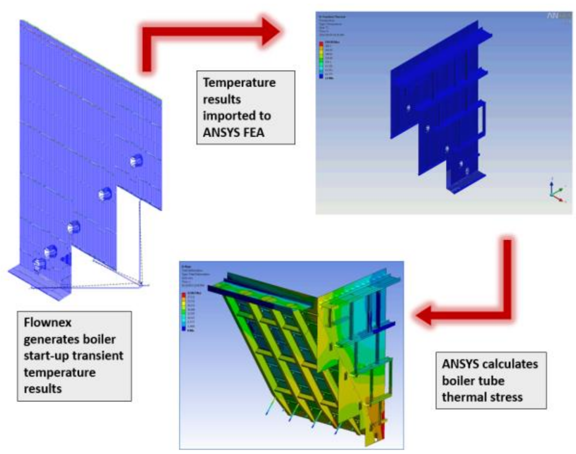


Figure 12: Flownex and ANSYS Mechanical link: Thermal stresses of a boiler.

7. Conclusion

Coupling 3D and 1D codes, the overall computational effort is reduced while the 3D fundamental modelling aspects are maintained at the same time.

It took nearly 20 years to have an efficient environment for such computation. At the beginning, the numerous software editors slowed down the creation of individual links, the second step often involved a third-party tool, but the recent merging of some software vendors shorten the collaboration.

Following this approach, several 1D tools are now linked with 3D tools in a unique simulation environment.

Now the coupling is completely automatic and be can used for sensitivity analyses to improve the system design.

Simulations can be optimised for both accuracy and minimization of run time, providing engineers with a robust set of simulation tools that meet industrial demands.

8. References

Bayrasy, Pascal (2010). *Generic Coupling of 1D System Codes with 3D CFD tools by MpCCI*: NAFEMS Seminar "Coupling 1D and 3D CFD: The Challenges and Rewards of Co-Simulation" Gaydon (UK).

Scholz, Dominik (2010). *Coupling to 1D Codes Amesim & GT Power and examples of 1D-3D couplimg*: NAFEMS Seminar "Coupling 1D and 3D CFD: The Challenges and Rewards of Co-Simulation" Gaydon (UK).

Jonnavithula, Sreenadh (2010). *1D-3D coupling with STAR-CCM+*: NAFEMS Seminar "Coupling 1D and 3D CFD: The Challenges and Rewards of Co-Simulation" Gaydon (UK).

Kelsall, David (2010). *Using Co-Simulation Middleware to Improve Aircraft Environmental Control Systems*: NAFEMS Seminar "Coupling 1D and 3D CFD: The Challenges and Rewards of Co-Simulation" Gaydon (UK).

ESTECO (2019). *Corporate website: www.esteco.com*

Mazzoleni, Erik (2017). *Flownex Simulation Environment: fluid networks with ANSYS power*: Newsletter EnginSoft. Pages 43-44.

MENTOR GRAPHICS (2019). *Corporate website: www.mentor.com*

Croegaert, Mike et al (2019). *A Novel Approach to Combining 1D and 3D Simulations to Accurately Model the Immersion Cooling of CPUs and GPUs of a Blade Server.*: NAFEMS WOLRD CONGRESS, Quebec City (Canada).

Laurens, Roelf (2017). *Boiler Tube Failure Thermohydraulic Analysis.*: Eskom-Flownex Case Study.