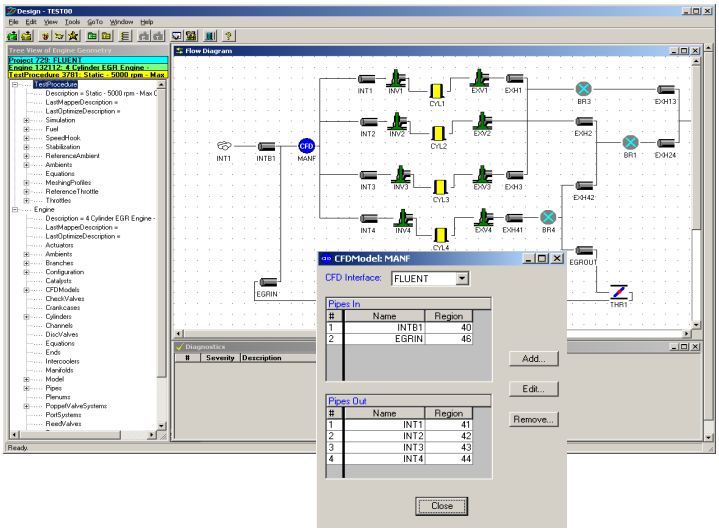
1D/3D CO-SIMULATION OPTION (FLUENT)

OPTIMUM Power Technology are excited to be able to offer a co-simulation option with FLUENT, a leading computational fluid dynamic (CFD) product, in VIRTUAL ENGINES v5.0. The benefits of coupling 1D engine cycle simulation models with 3D computational fluid dynamic (CFD) models are well documented and include:

- Improved representation of multi-dimensional components such as air boxes in 1D model
- Improved boundary condition data for 3D model
- Interaction between 1D and 3D models for more integrated design process

Senior developers at OPTIMUM Power Technology and its research partners have many years experience in coupling 1D and 3D models. Using this experience, OPTIMUM has been able to create a powerful yet easy-to-use interface between these two state-of-the-art simulation environments.



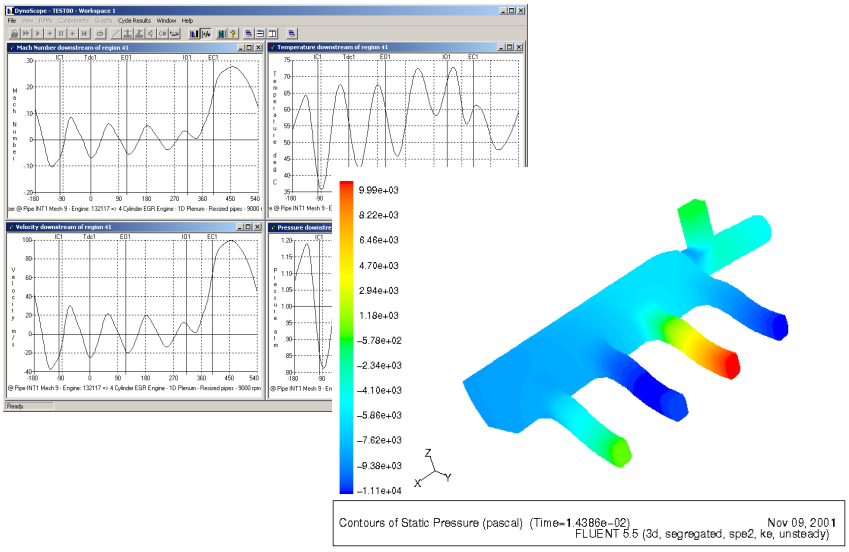
CFD component definition in VE

The 1D/3D co-simulation option includes a single custom library that is incorporated within the FLUENT model. No other changes are required to either product. The VIRTUAL ENGINES (VE) model is assembled as normal through the engine design interface. A CFD component, which allows up to 20 1D pipes to be connected to the CFD interface, can be placed within the 1D model. The only user-specified parameters defined in the model are the corresponding FLUENT region numbers or zone ID's. Error handling embedded within the custom library ensures that these match those in the CFD model at runtime. Prior to co-simulation the user submits the engine model with a corresponding test-procedure. The model is then queued for simulation.

Within FLUENT, the user specifies the appropriate user-defined function (UDF) for each boundary condition to be solved. The custom library provided by OPTIMUM Power Technology contains all the required UDF's to solve each FLUENT boundary type.

When co-simulation is required, the user executes the FLUENT model. The FLUENT model acts as a server application and attaches the VIRTUAL ENGINES solver as a client. A check is made to ensure that a CFD component with the corresponding region numbers exists in the 1D model. Communication then commences directly through memory on a variable time-step basis controlled by VIRTUAL ENGINES ensuring efficient data transfer.

An example of co-simulation is shown below. An intake manifold for a 4-cylinder 4-stroke engine incorporating exhaust gas recirculation (EGR) has been created in FLUENT. The full engine model is defined within VIRTUAL ENGINES and a pipe is connected to each of the appropriate FLUENT regions using the CFD component. Co-simulation at 9000 rpm demonstrates the impact on both models.



Analysis of coupled VE-FLUENT model