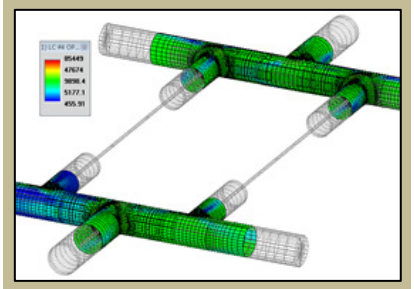
FE/Pipe

Finite
Element
Analysis

Pressure Vessel &
Piping Engineers



ASME Sec. VIII Div. 2 2007 Edition, A08 Addenda

Fast, Accurate and Easy to Understand Results

Template-Based Design Tools

Code Compliance Reports

Today's Engineer requires state-of-the art tools and design techniques in order to accurately and realistically model the interworkings of a plant or system.

Finite Element Analysis provides the right answer.

What separates FE/Pipe from the rest of the general FEA tools?

The ability to rapidly construct PVP geometries and produce ASME stress reports sets us apart. FE/Pipe makes modeling typical pressure vessel and piping geometries faster and easier than is possible with general FEA tools due to the parametric design philosophy. The parametric approach used in FE/Pipe permits even novice finite element analysts to construct accurate models using only dimensional input. FE/Pipe automatically creates the model geometry, element mesh, applied loads, and boundary conditions based on standard dimensions.

Results are presented in terms of ASME requirements instead of generalized stress results.

The FE/Pipe focus is on nozzles, supports, flanges and pipe. The program maintains some heat transfer capability and elastic / plastic analysis with large rotation.

Why should I use FE/Pipe?

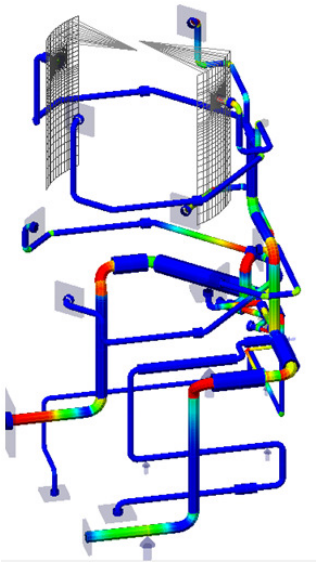
FE/Pipe addresses many needs of PVP engineers working in today's market. The following are a few of the most important reasons to use FE/Pipe:

Typical pressure vessel design codes, such as ASME Section VIII, cannot address all design cases. For instance, external loads on nozzles are not addressed within the Code. In such cases, engineers need to go outside of the Code and apply recognized design procedures such as finite element analysis using FE/Pipe.

Simplified calculation methods commonly used in the PVP industry such as WRC 107/297 are based on limited test data and are known to be inaccurate in many cases. FE/Pipe provides realistic answers for all cases.

FE/Pipe has been designed to meet the needs of the PVP industry. General FEA tools are not tailored to the PVP engineer.

FE/Pipe produces ASME code output reports in accordance with ASME requirements.



FE/Pipe automatically produces stress intensification factors and flexibilities for typical piping junctions. These are weak points in the piping code and FE/Pipe can be used to supplement detailed piping analysis.

What is included with FE/Pipe?

FE/Pipe purchase includes the PRO series of products:

NozzlePRO • AxiPRO • MatPRO

FE/Pipe service plan includes the FE series of products:

FE107 • FESIF • FE661 • FEBend

…and the ***PCL-Gold Pipe Stress Module***

FE/Pipe

info@paulin.com

MODELING - from the complex to the sublime...

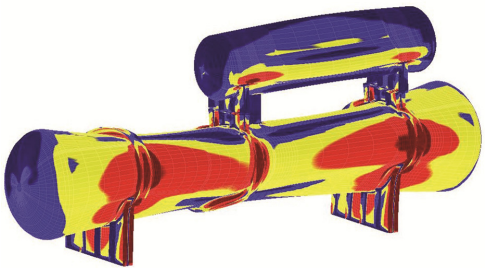
A large number of common PVP geometry can be modeled using FE/Pipe templates. Standard models routinely analyzed by our users include...

- Unreinforced and reinforced tees, lateral, and hillside nozzles or piping intersections
- Saddle supported vessels and heat exchangers
- Large diameter piping and ducting using shell elements
- Piping systems using standard 6 degree of freedom (DOF) beam elements, new 18 DOF beam elements, or shell elements
- Vessels with all geometric features including nozzles, support skirts, heads, structural clips, and stiffening rings.
- Flanges with bolt loads, pressure, external loads, and thermal analysis
- Tangential entry nozzles in cylindrical shells (rectangular, obround, and cylindrical nozzles)

LOAD ANALYSIS

FE/Pipe includes a load case processor that automatically accounts for load cases that contribute to failure in piping and pressure vessel components.

- Weight, Operating, Occasional, Thermal and Pressure
- Internal or external pressure
- Applied point or surface loadings
- Piping loads applied to nozzles
- Wind
- Acceleration due to ship motion or transportation
- Seismic
- Fluid head



SOLUTION CAPABILITIES

FE/Pipe offers similar solution capabilities that would be found in any FEA tool.

- Element library (beam, shell, axisymmetric, & brick)
- Linear elastic analysis
- Material non-linear analysis (plasticity)
- Dynamic/modal analysis
- Dynamic harmonic analysis
- Eigenvalue buckling
- Steady state and transient thermal analysis
- Stress stiffening (large displacement)
- Refractory
- 2007 Edition of ASME Section VIII-2

PCL-GOLD PIPE STRESS MODULE

FE/Pipe clients who maintain active support service agreements receive access to the PCL-Gold.

- Automatic Fatigue Damage Calculations for multiple load cases
- Path dependent, convergent friction algorithm
- Refractory lined pipe
- Glass lined pipe
- Hinged expansion joints with friction
- 07-02 i-factor and k-factor modeling (P. Vu paper)
- FEA i-factors and k-factors
- Pressure fatigue
- i-factors and k-factors for flat, conical, elliptical, spherical and dished heads
- Ability to enter 10-SIFs for branch connections
- Superelement results
- Structural elements & pipe elements in same interface

RESULTS VERIFICATION

At the most basic level, the element formulations and related output have been compared against classical hand calculations. Additionally, FE/Pipe has been benchmarked against other general FEA software tools. The benchmark problems have included a range of complexity from single element verification problems to complete PVP analysis cases.

In addition, Paulin Research Group routinely conducts experimental work in the PRG Laboratory. FEA models are constructed for comparison against strain gauge measurements from experimental cases. This work extends to included burst tests, fatigue tests, cryogenic work, and heat transfer experiments. Further, PRG is active in the PVP research field and continually processes other available test data for validation of the model building and analysis approaches used in FE/Pipe.