Getting Inside the Power Cylinder

An integrated workflow to improve quality and lead time in piston and liner CAE analysis

Contents

- **Introduction**
- General workflow
- Thermal analysis
- Piston dynamic analysis
- Piston stress analysis
- Liner stress analysis

Getting inside the power cylinder
Introduction

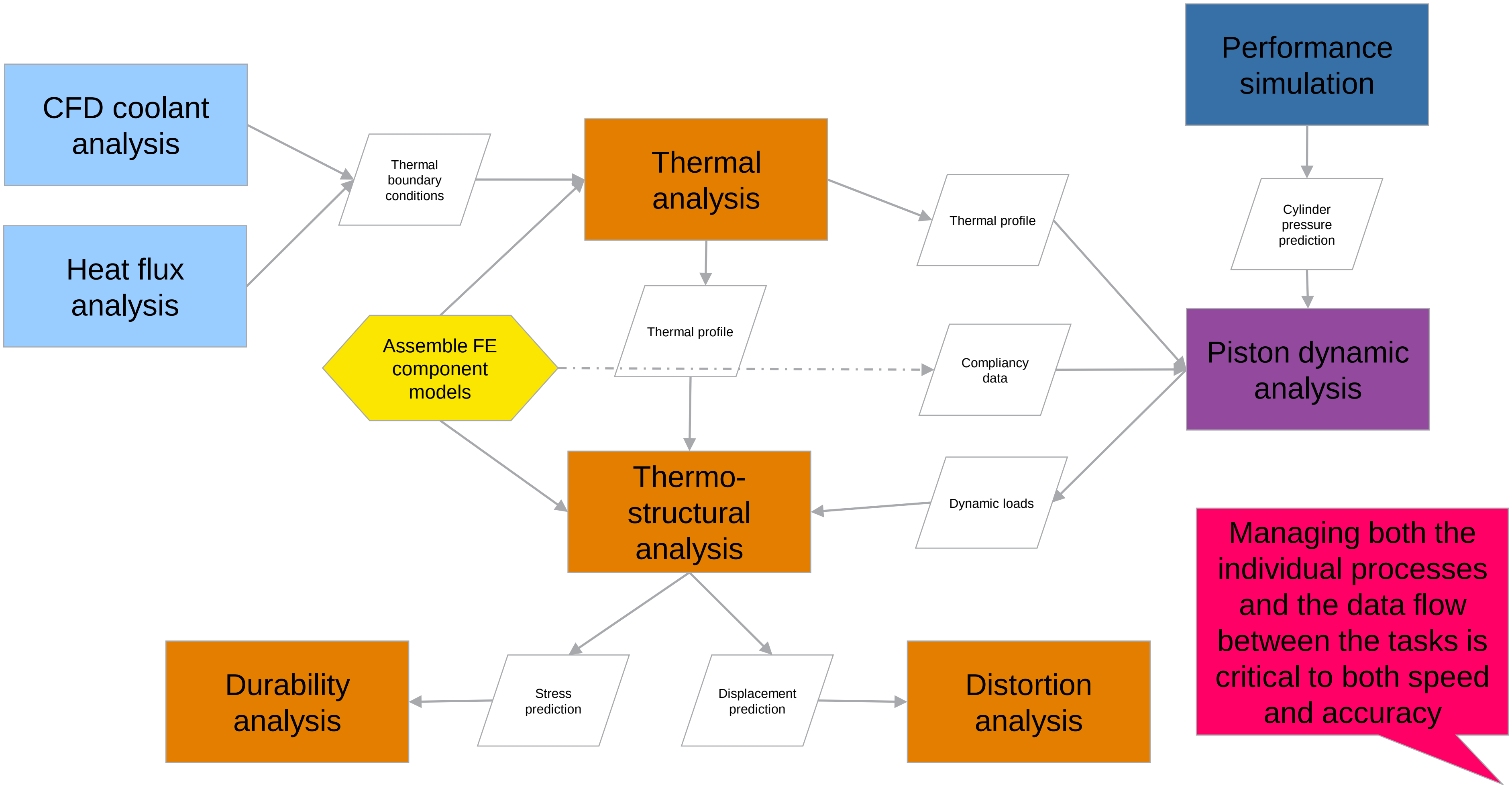
- This paper presents an integrated workflow and toolset developed by Ricardo to manage the different tasks required for a comprehensive analysis of the piston and liner assemblies in an internal combustion engine
- It shows how software development has enabled the engineer to successfully understand the thermal, dynamic and structural behaviours of the system and so use CAE to drive design
- It also presents how Ricardo have combined their individual analysis tools to enable the engineer to easily manage each separate task and the associated data flow, thus improving the reliability of the results and greatly reducing analysis lead time
- For brevity the paper focuses on the detail of the finite element solutions in the process and shows how the FEARCE tool has been a major asset to the CAE activities at Ricardo
- The paper uses a Ricardo single cylinder engine to demonstrate the process that has been used successfully on a number of recent client projects

Contents

- Introduction
- **General workflow**
- Thermal analysis
- Piston dynamic analysis
- Piston stress analysis
- Liner stress analysis

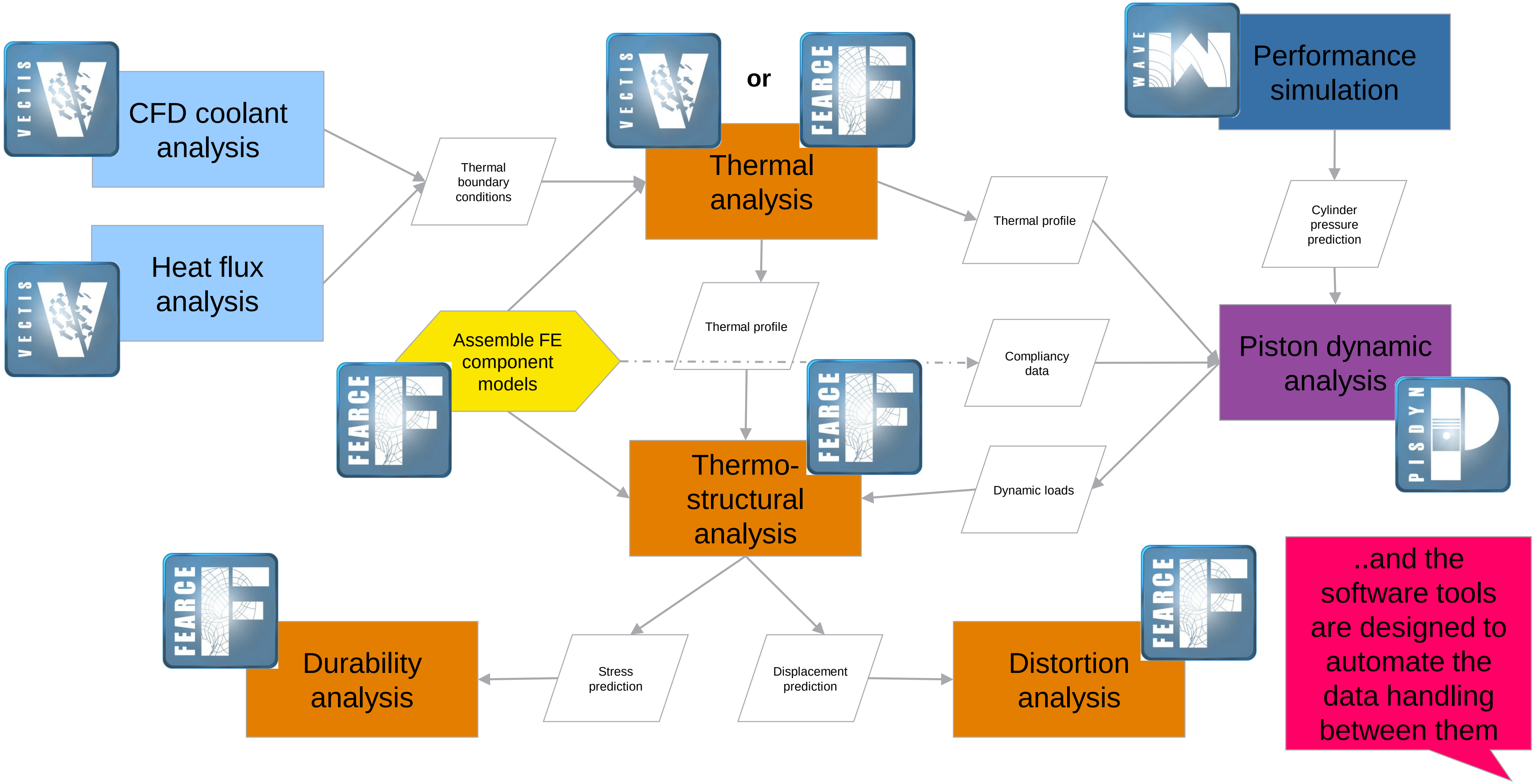
Getting inside the power cylinder
General workflow

- In order to perform comprehensive studies of the piston and liner assemblies the following tasks need to be carried out



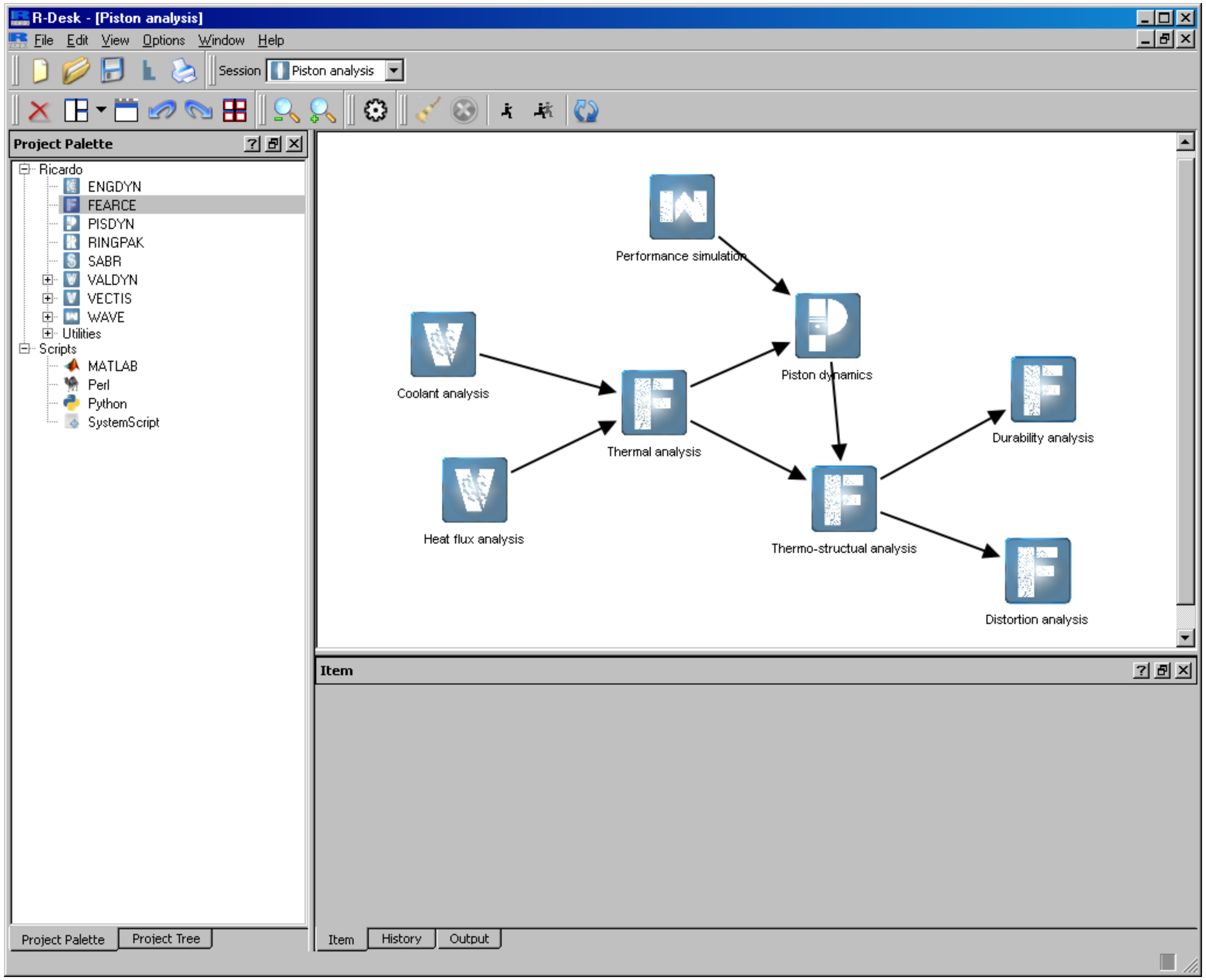
Getting inside the power cylinder
General workflow

- Ricardo software have products to perform all of the key processes so the workflow becomes



Getting inside the power cylinder
General workflow

- …and this workflow can be set up within a Ricardo Project as simply as shown



- Each individual analysis is directly linked to the next process in the flow
  - The Project tool then controls the dependencies
  - As such the engineer can both define and execute the complex data flow through one simple tool
- At this stage the Project acts as a template allowing a process to be clearly defined and shared between analysts
  - Our job is then to simply build the component solutions!

Contents

- Introduction
- General workflow
- **Thermal analysis**
- Piston dynamic analysis
- Piston stress analysis
- Liner stress analysis

Contents

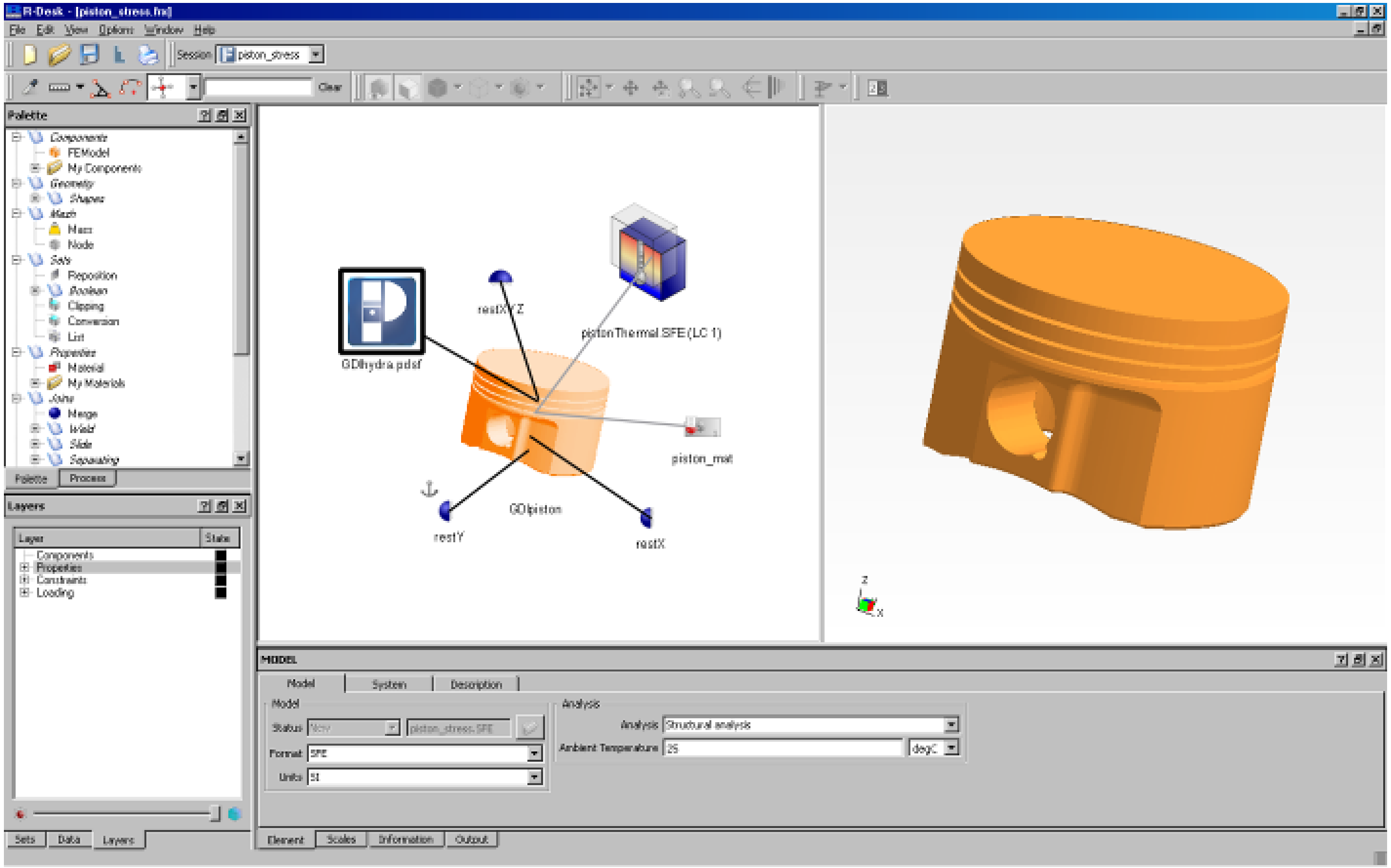
- Introduction
- General workflow
- Thermal analysis
- Piston dynamic analysis
- **Piston stress analysis**
- Liner stress analysis

Piston stress analysis
Methodology

- All of the preparation analyses have now been completed and we now have a full set of boundary conditions available for a thermo-mechanical solution
  - Thermal loads through VECTIS CFD and FEARCE FE analysis
  - Mechanical loads through PISDYN
    - Plus cylinder pressure from WAVE performance simulation
- We now need to combine these loads and apply them to a finite element solution to predict distortion, stress and durability through the cycle
  - To do this we perform what is known as a quasi-static analysis where we apply loads at distinct crank angles within the cycle and combine these to give a representative duty cycle to predict durability
- This requires the application of numerous complex load profiles across multiple crank angles and so is a very significant challenge in data and time management
- The tools within FEARCE to link directly to the external data sources and automate the load application provide a significant aid in this endeavour

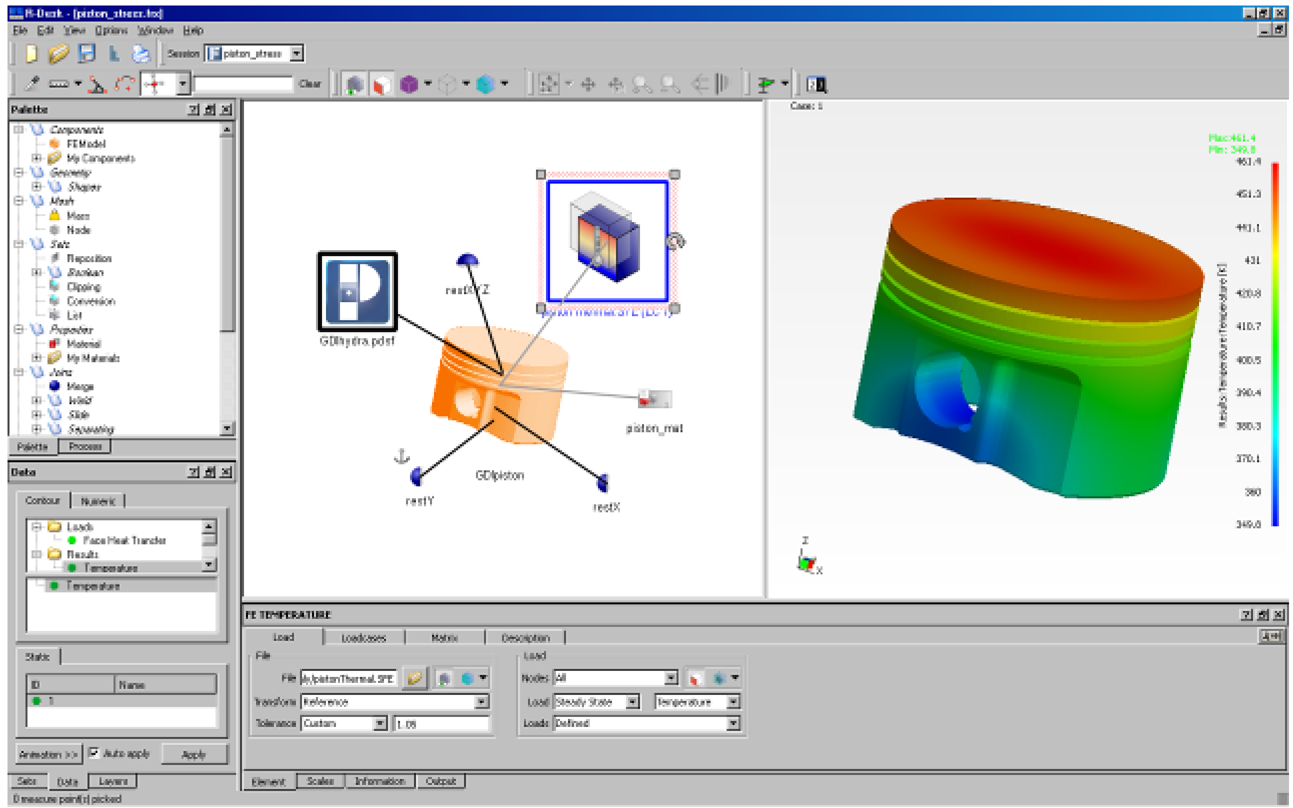
Piston stress analysis
Applying loads for the stress analysis

- Tools within the FEARCE code allow us to link directly to the results files containing the temperature and mechanical loads



Piston stress analysis
Applying temperatures

- The interpolation tool links to the FE models that were generated from the thermal analysis
  - These can be different meshes or even geometries to the one we are using for the structural analysis
- FEARCE interpolates the nodal results from the donor model onto the mesh being used for the structural analysis
- The loads can be mapped to all or part of the model depending upon the requirements of the analysis
- Multiple loads can be interpolated and even combined

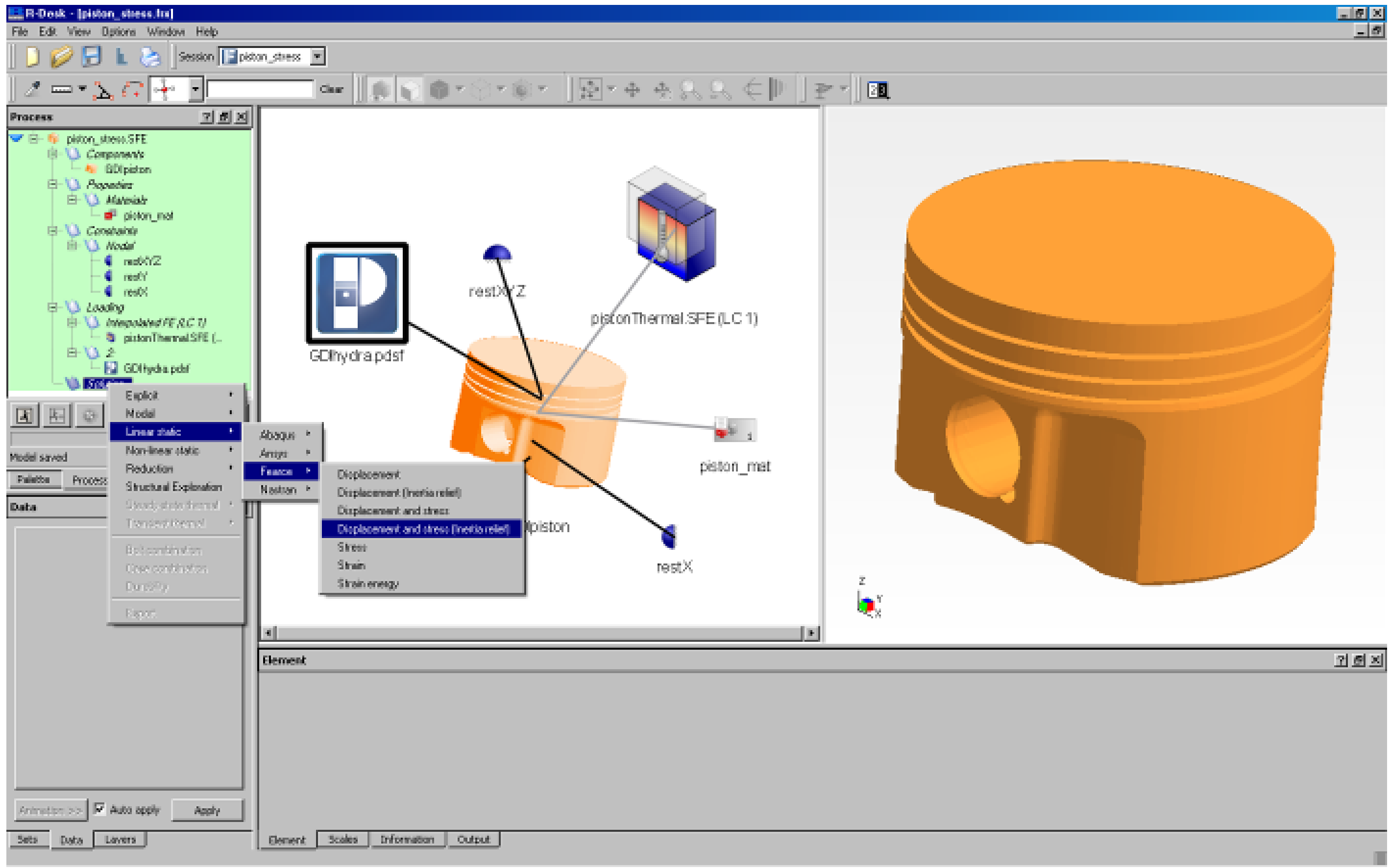


Piston stress analysis
Applying mechanical loads

RICARDO

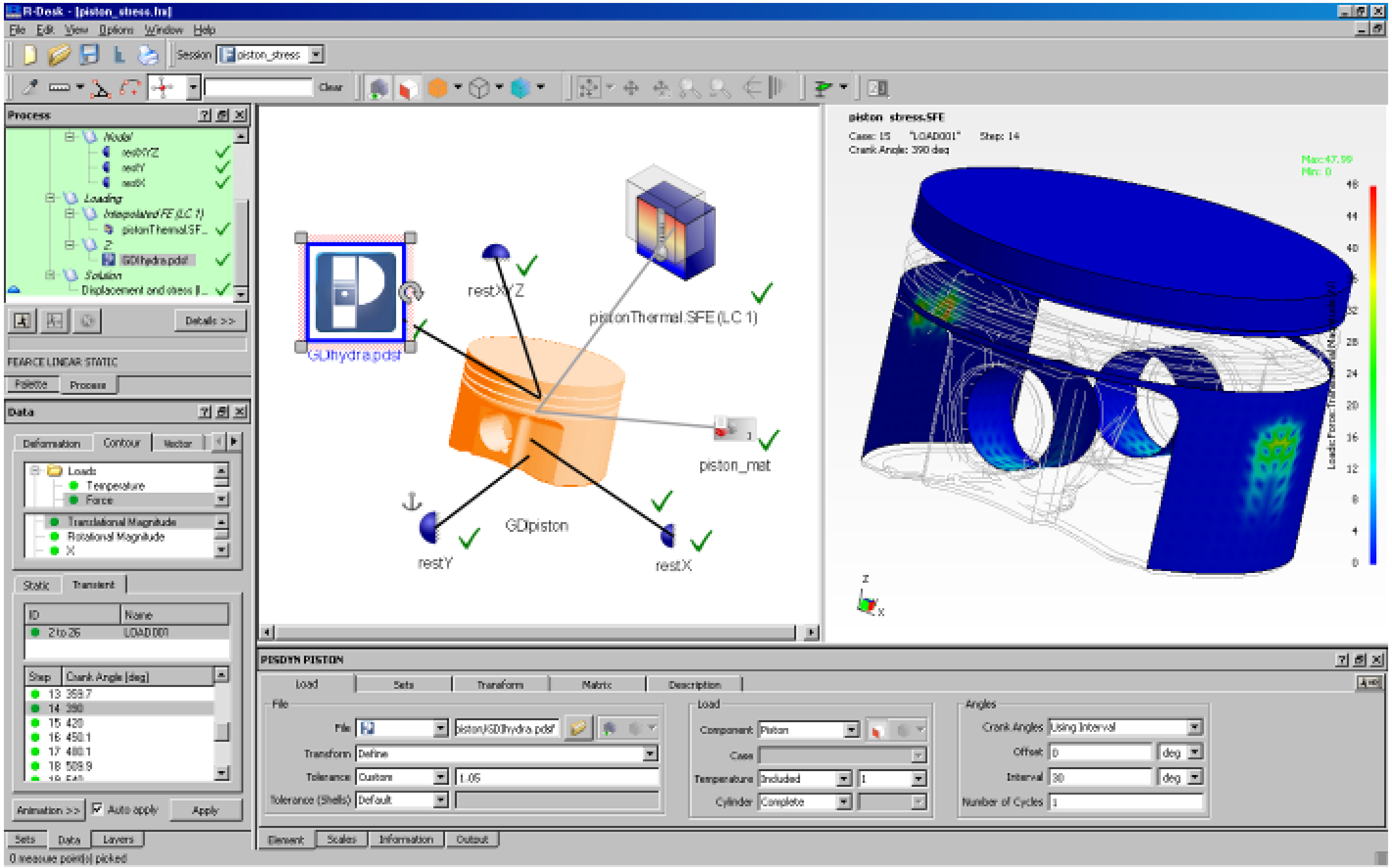
- Any assembly loads can also be applied to the FE model using FEARCE
  - For the piston this could be interference fits of ring inserts or bearing shells
- Restraints and constraints can also be added to the model so that the system loads can be reacted against

For the piston analysis, inertia relief can be added in the solution and FEARCE can set this up for either its own solver or similar solver methods in ABAQUS and ANSYS



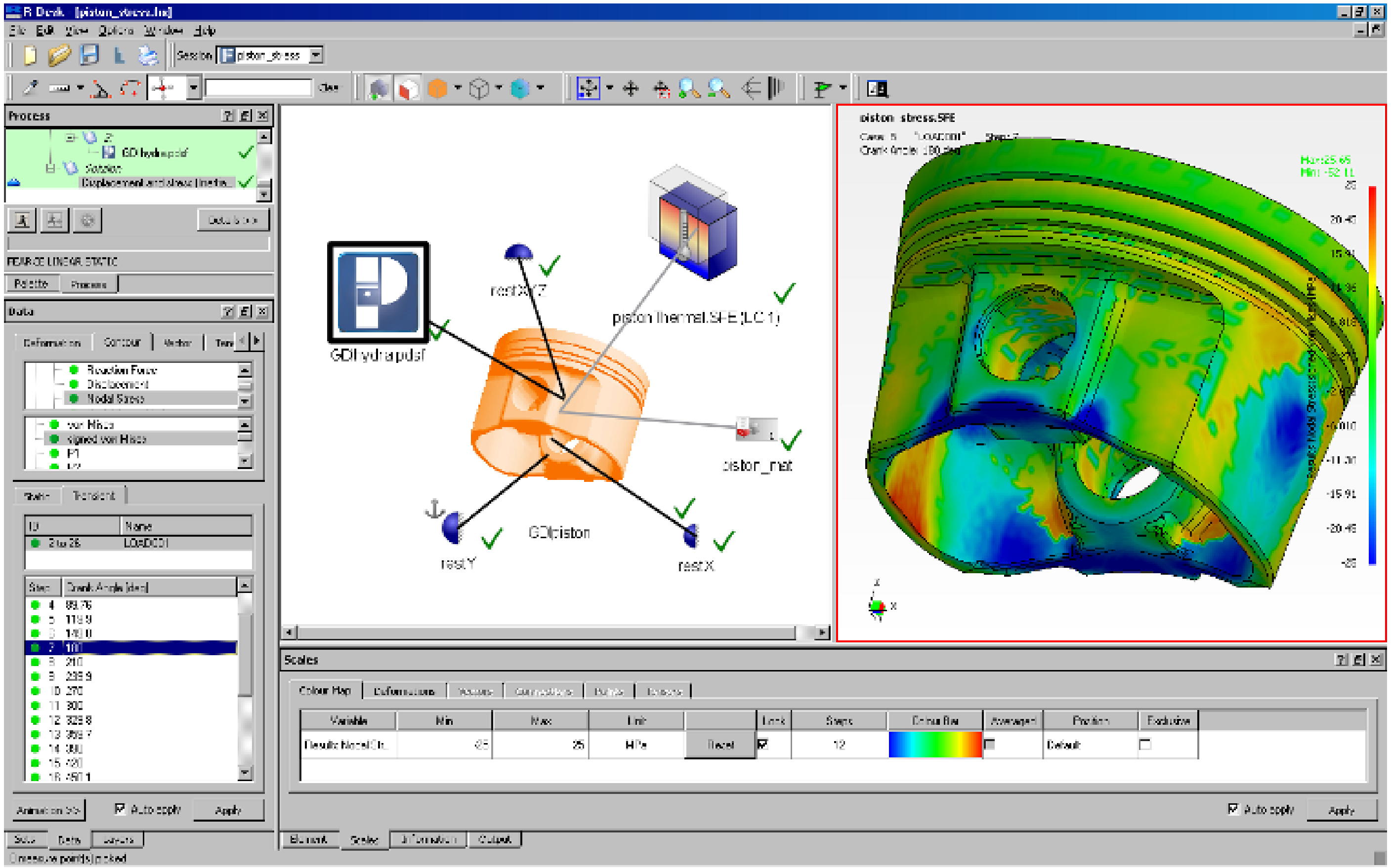
Piston stress analysis
Applying mechanical loads

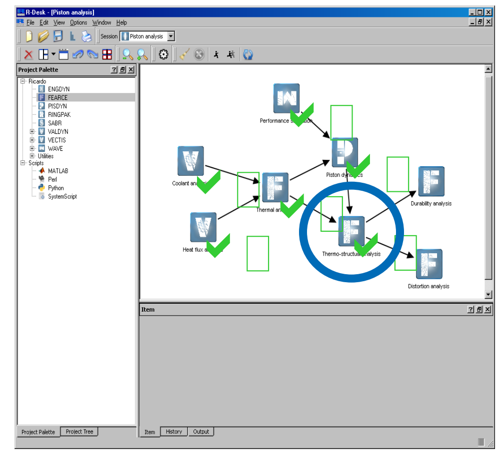
- Once the set-up is completed, FEARCE reads the external model data files and automatically applies the loads to the system for each crank angle case
  - It then either solves directly or creates an input deck for a 3rd party solver



Piston stress analysis
Stress solution

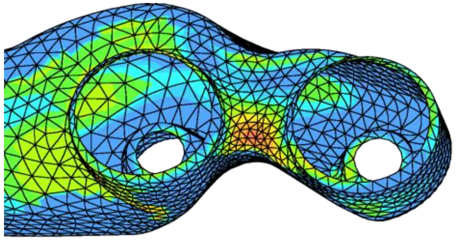
- On completion of the solution a full set of stress and displacement results are available at each chosen crank angle across the cycle



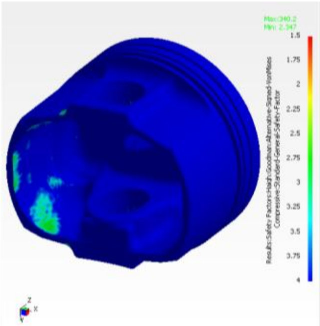


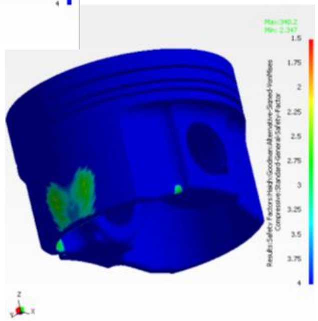
We have now completed the process up to the thermo-mechanical simulation

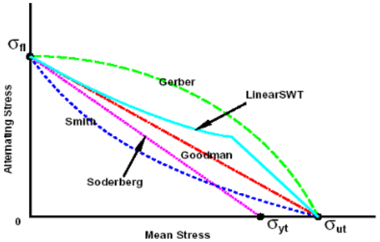
Piston stress analysis
Durability analysis

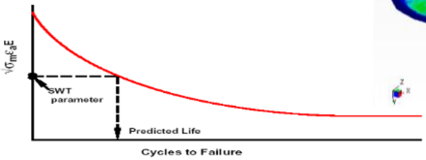


- Now that we have stress predictions across the engine cycle we can use this data to perform a durability analysis of the component
- FEARCE has its own durability solver that has been used extensively by Ricardo for over 20 years
  - The solver contains numerous linear and non-linear algorithms including;
    - Haigh (Goodman, Gerber, Smith and Soderberg curves)
    - Brown-Miller
    - Crossland
    - Dang Van
    - SWT

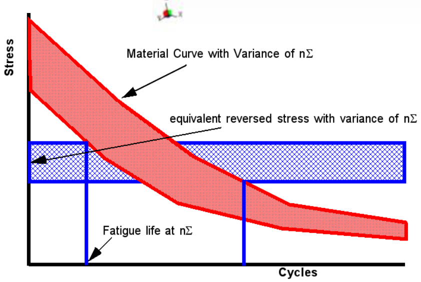






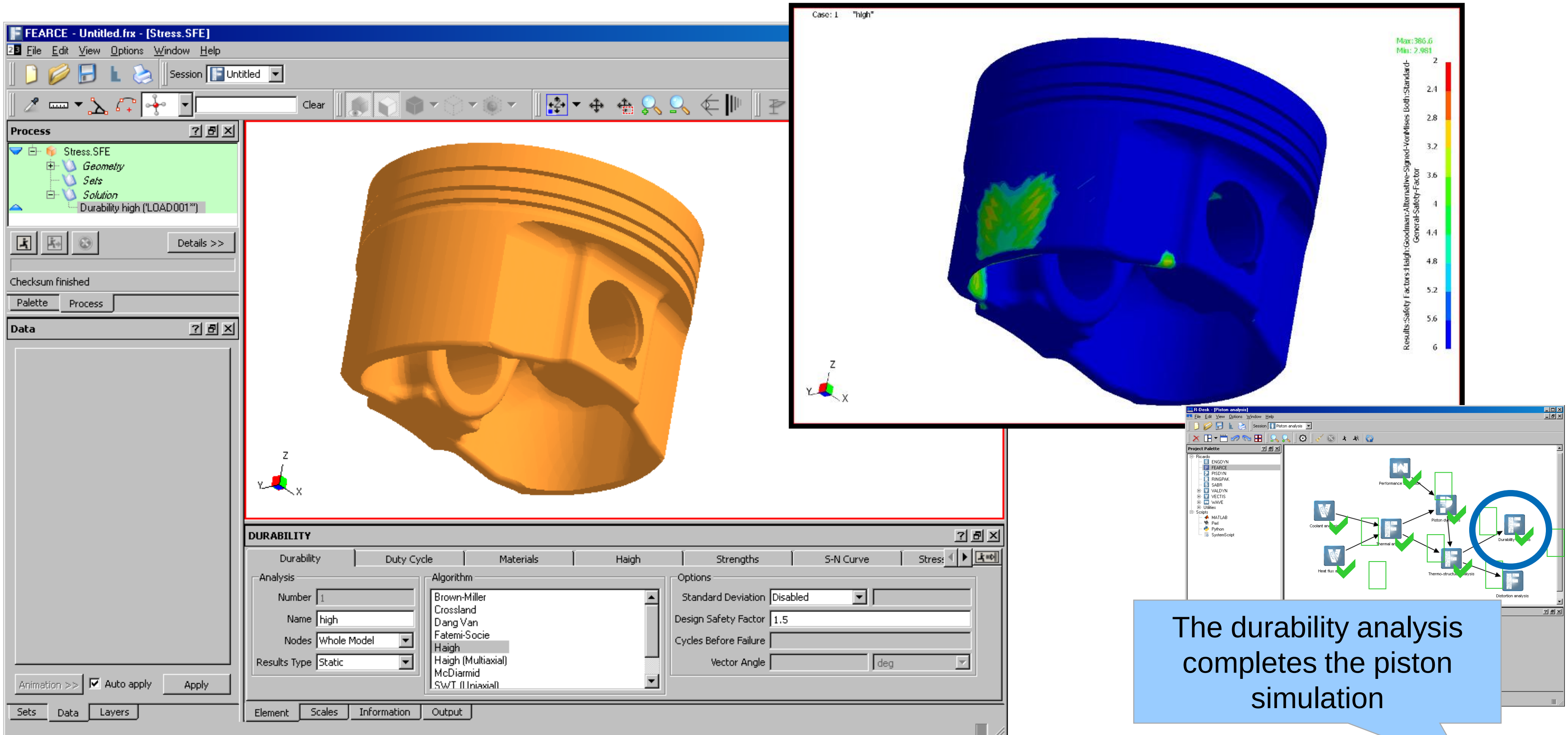


  - The solver also provides a number of alternate equivalent stress options as well as stress correction methods to account for such phenomena as high ductility
- With 8.0 the solver was updated to include reliability analysis
  - With which a user can provide material data with statistical variation to obtain a confidence level of the final result



Piston stress analysis
Durability analysis

- The durability solution is easily set up within the GUI
  - ..and the solution time is only a few seconds



The durability analysis completes the piston simulation

Contents

- Introduction
- General workflow
- Thermal analysis
- Piston dynamic analysis
- Piston stress analysis
- **Liner stress analysis**

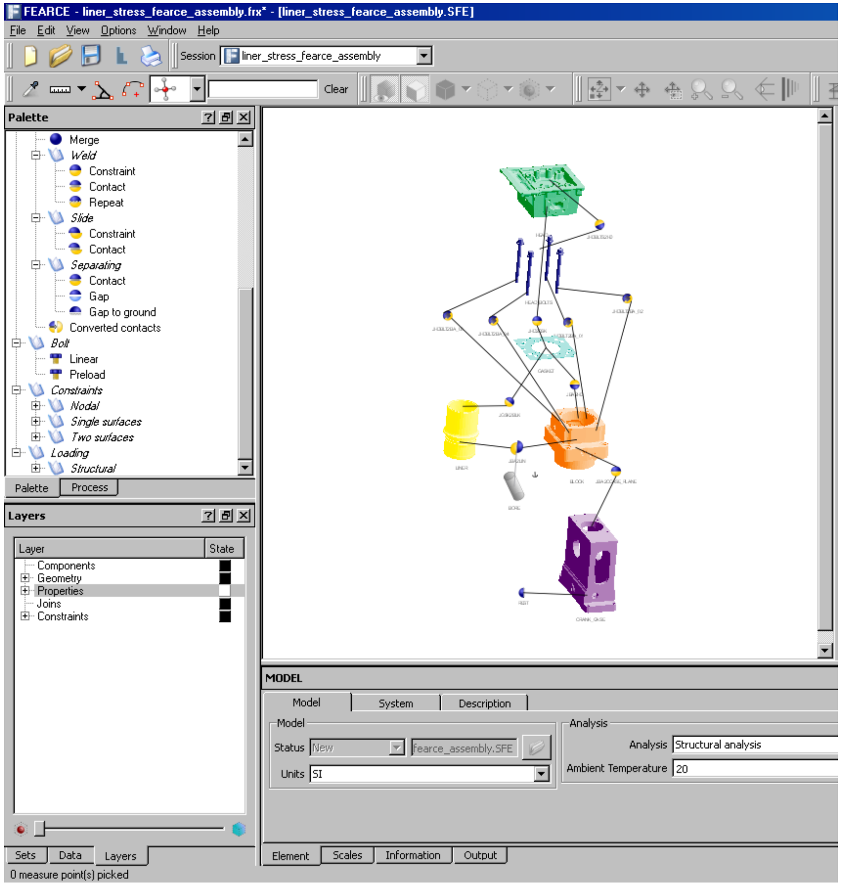
Liner stress analysis
Methodology

- For the stress analysis of the liner we need to use the powertrain assembly
  – For this reason we also have assembly loads to add to the thermal and mechanical loads we used for the piston analysis
- The main assembly loads come from the cylinder head bolts
  – These have a very large contribution to bore distortion so it is important that the effects are modelled accurately
- As with the piston, we also have to combine these loads with the mechanical forces on the liner predicted by PISDYN
  – Once again, FEARCE provides us with the tools to apply all of the loads required for the liner analysis

Liner stress analysis
Assembling the model

- The physical joins between the components are defined in the 2D network by simply choosing the appropriate type of join and attaching it to the mating surfaces

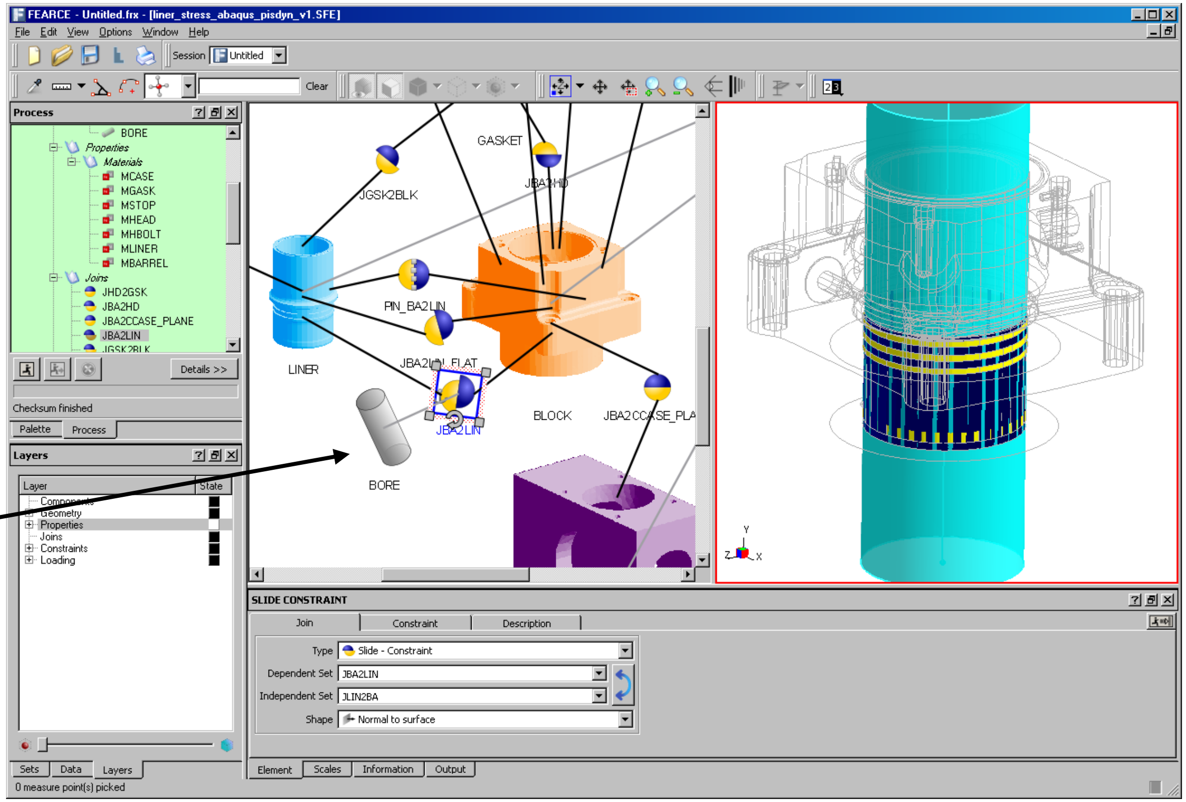
- FEARCE also contains automated tools for the joining of non matching meshes between component models
  - No reliance on matching nodes or pre defined node numbers
    - User simply specifies the surfaces to be joined
- Numerous types of joins can be generated including
    - Weld, slide, contact, thermal and structural gaps
  - Automatic checking of joins with conflicts, fixes and warnings flagged



Liner stress analysis
Assembling the model

- The two mating surfaces don't need to have matching mesh or even be of the same extent
  - Here the liner surface in yellow is only partially covering the block side surface
    - FEARCE will only create appropriate joins for those nodes within tolerance

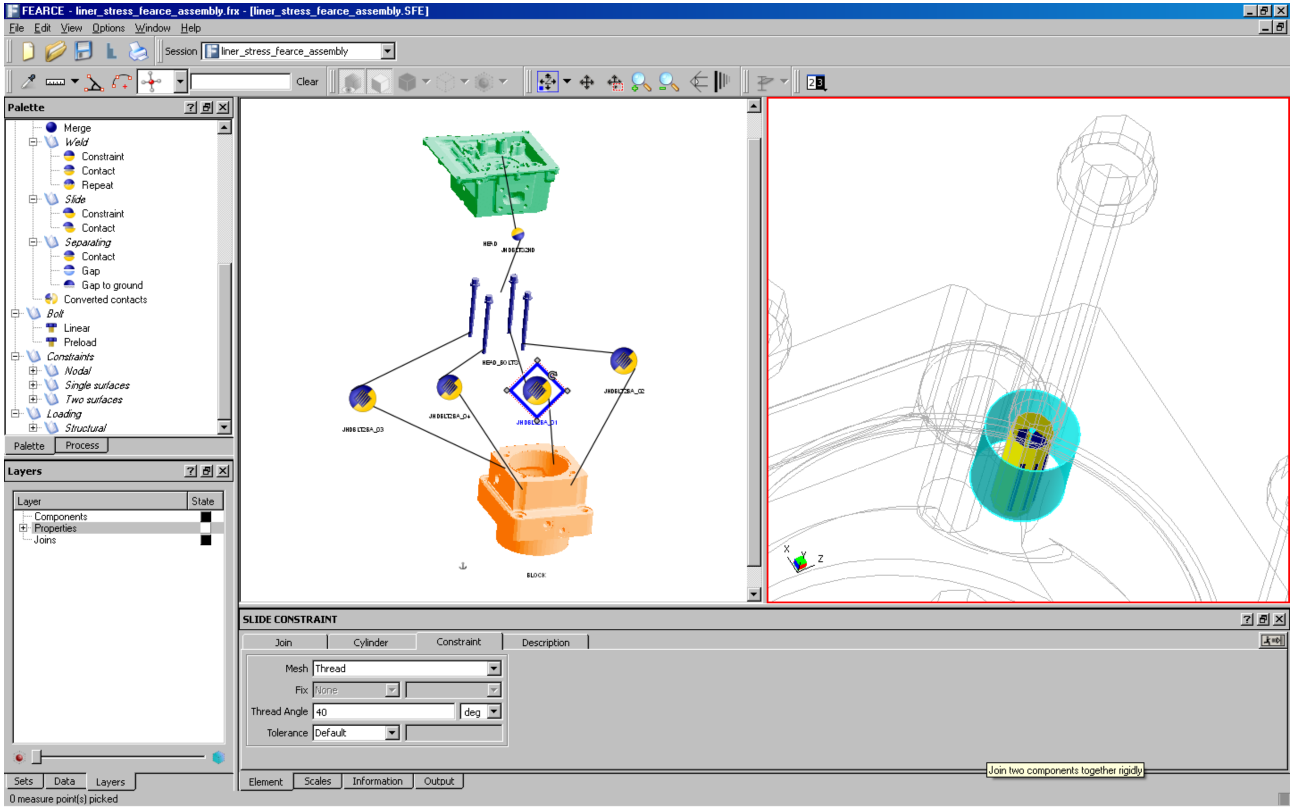
- Geometric shapes can also be defined to provide surfaces over which degrees of freedom are allowed
  - In this case a cylinder shape defines the sliding freedom of the join



Liner stress analysis
Assembling the model: Adding bolts

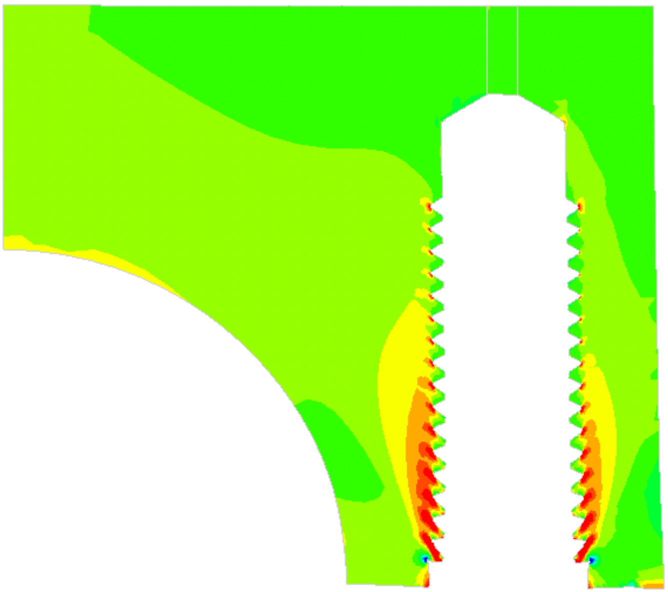
- Included in the options is a tool for accurately applying bolt loads through the threaded region
  - A simple cylindrical geometry can be used for the bolt thread region and the thread angle is defined as an input to the join

- The method uses a linear join so does not add to overall solution time
- The join is created by simply linking the element to the mating faces and adding the thread angle to the panel
- Bolt loads can be added as either thermal shrinkage or preloads

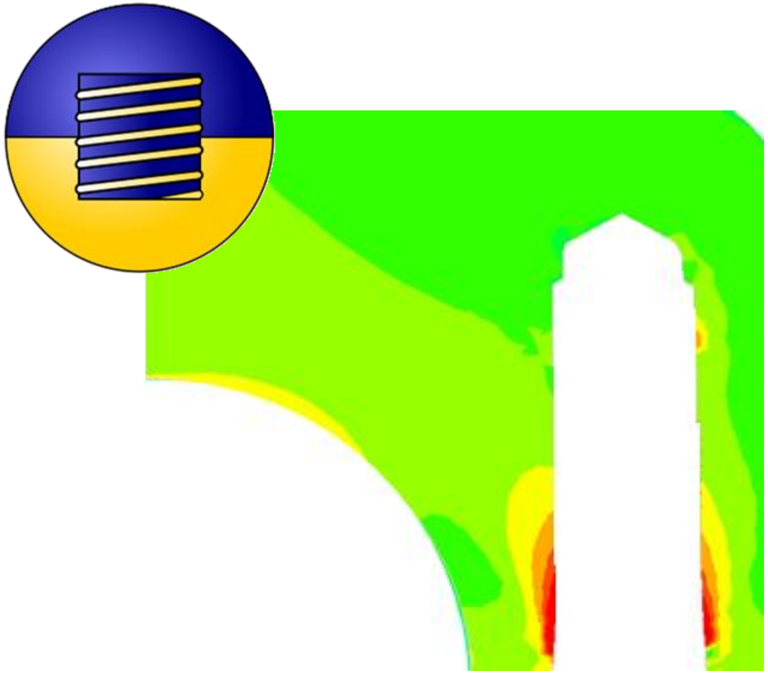


Liner stress analysis
Assembling the model: Adding bolts

- The method has been correlated against full non-linear analyses of threaded bolt models
- Results show a very good match



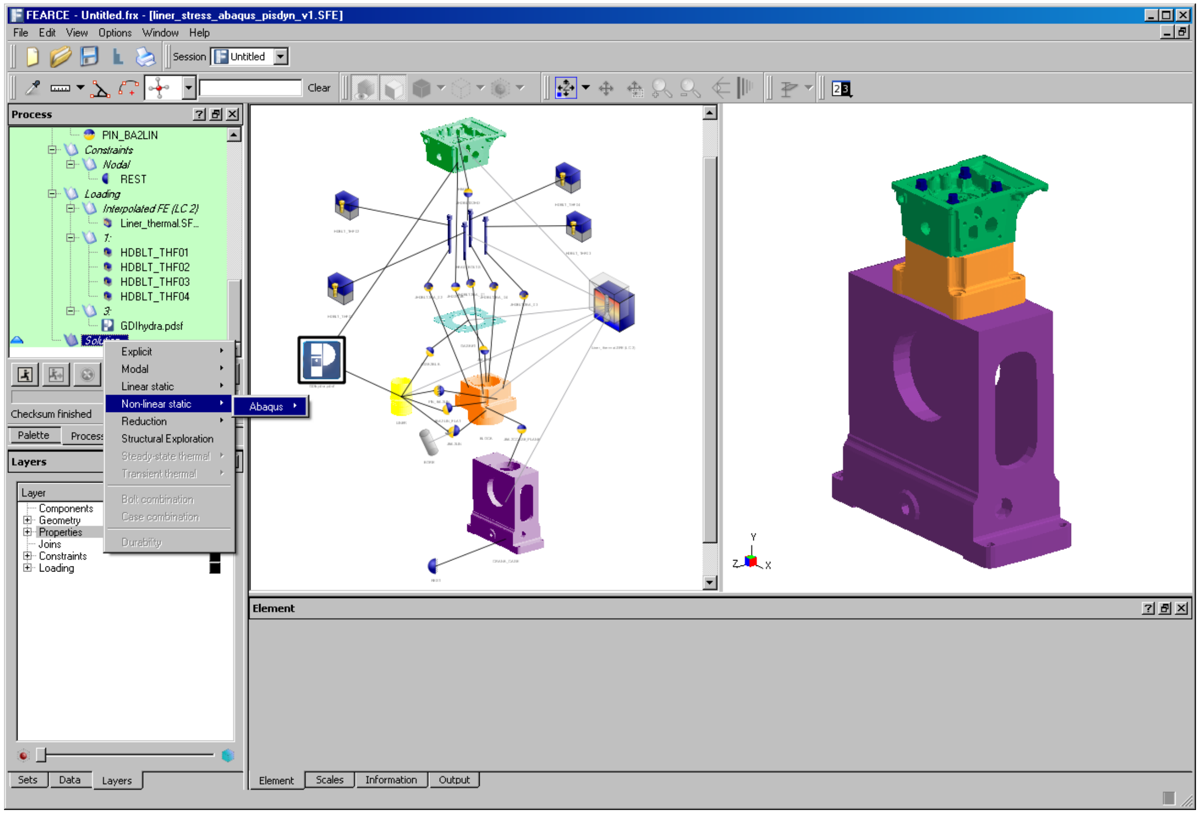
Non linear analysis including thread geometry and contact friction

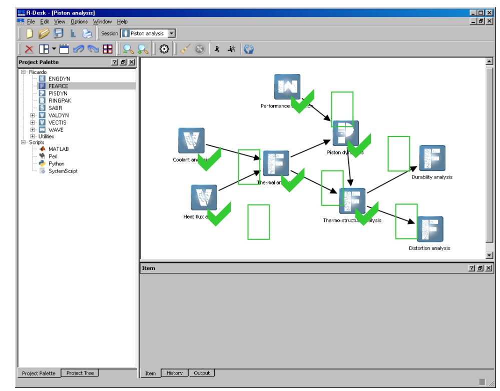


FEARCE linear bolt constraint using cylindrical geometry, non matching mesh and user defined thread angle

Liner stress analysis
Assembling the model

- So FEARCE allows us to quickly assemble our complex model
- We can apply the bolt loads as linear thermal reduction loads or as preloads
  - ..and as with the piston we can add loads from our previous mechanical and thermal analyses by simply linking them to the loading areas

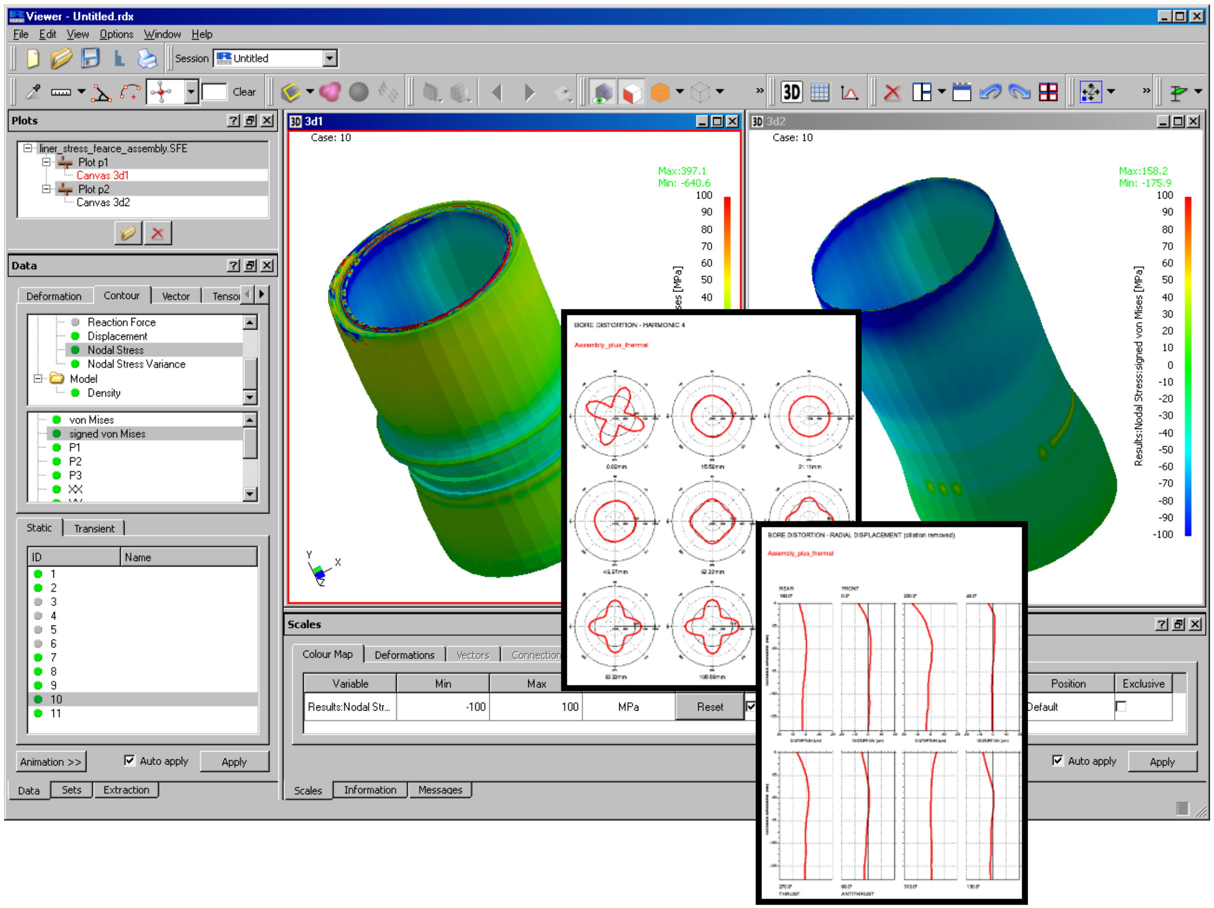




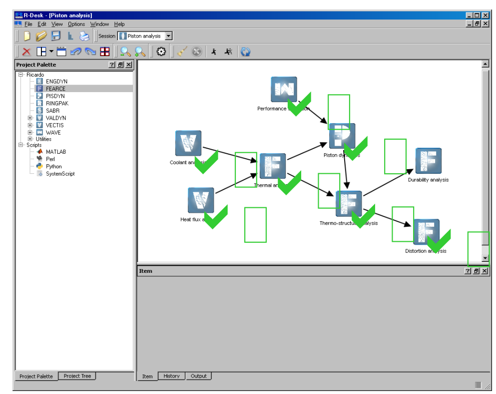
The stress analysis can be run just as before. Either directly in FEARCE or by setting up a 3rd party solver such as ABAQUS or ADVC

Liner stress analysis
Running the stress analysis

- The stress analysis can be run just as before
  - Either directly in FEARCE or by setting up a 3rd party solver such as ABAQUS, ANSYS or ADVC

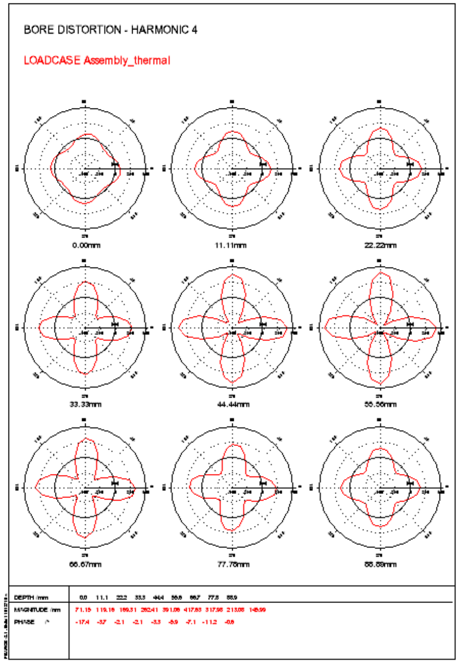


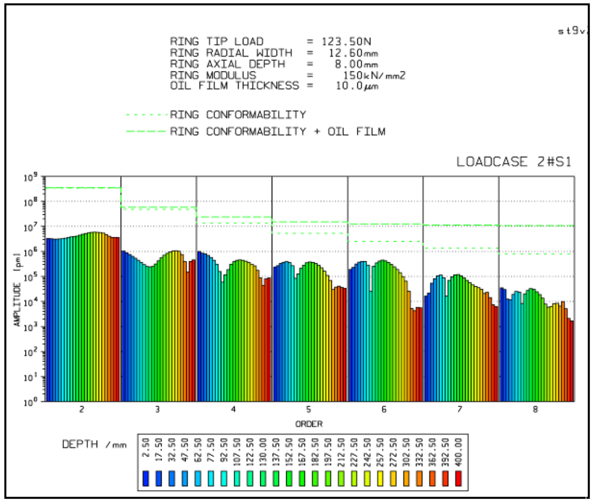
- FEARCE can easily generate the bore distortion harmonic plots from the results of the analysis
  - Typical run time is seconds!



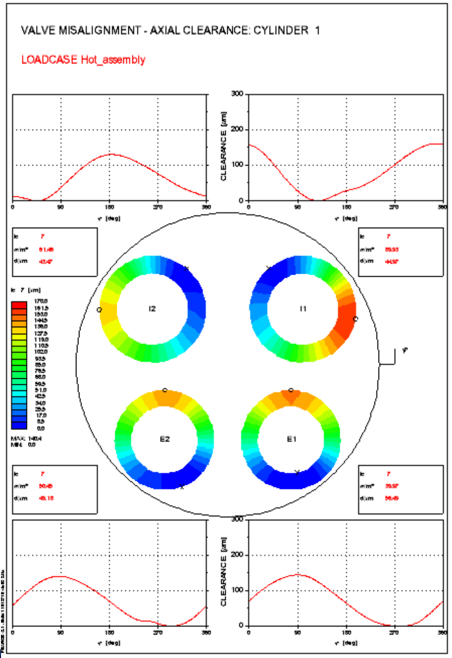
Liner stress analysis
Analytically assessing the distortion

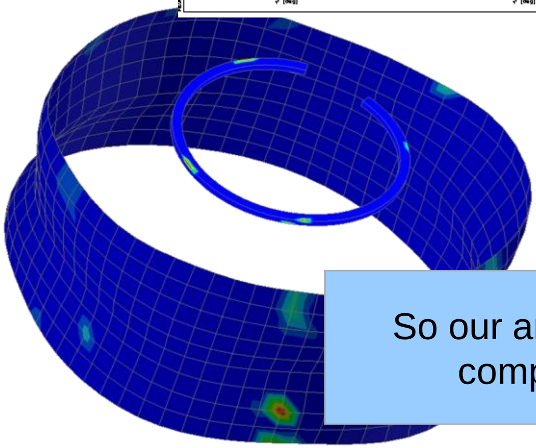
- FEARCE has numerous distortion analysis tools that allow the fast calculation of bore and bearing distortion – as well as valve guide misalignment
- Harmonic contributions and piston ring conformability are calculated automatically for the bore distortion





Harmonic piston ring conformance





So our analysis is complete!

SUMMARY

- Ricardo Software have a complete toolset for the thermo-mechanical simulation of powertrain systems
  - Capable of delivering results from initial boundary condition calculations right through to stress, distortion and durability analysis of the working components
- Ricardo Software’s integrated approach means that the complete process is managed by the tools themselves, ensuring accurate data transfer between the key stages and delivery of accurate results quickly
- Central to the process is the FEARCE software, which manages all of the complex tasks involved in the finite element analyses
  - From model assembly, through the application of boundary conditions from external packages, to the solution and further processing of results
- Real powertrain design at your fingertips