Case Study : Bradken Rail

Gaining significant productivity improvements through ANSYS Workbench

Expect huge productivity improvements with the new ANSYS Workbench environment

The use of the ANSYS Workbench environment at Bradken Rail has lead to significant reductions in the time and effort involved for routine product development and product validations compared to previous 'traditional' finite element validation techniques. The description below outlines how these productivity improvements were enabled.

Benefits

The benefits include reduced cost and development times. Also, the additional resources which become freed up can be dedicated to perform the more advanced studies. More complicated analysis types such as non-linear contact, optimisation, fatigue etc. are being tackled with a more routine approach because of the ease of use of the new interface. Analysis is no longer limited to a dedicated analyst with specialised skills. With minimal training, the CAD product designers can complete the analysis and the expertise is then required to review and interpret the results.

Products

One of the core products that Bradken Rail designs and manufactures are the bogies beneath freight wagons. These are made up of steel castings which must withstand large dynamic loads in a severe fatigue environment.

Traditional approach

Using the 'traditional' FEA approach, a CAD model would be imported to the analysis package, meshed, loads and boundary conditions applied. The model would be solved and the results reviewed. Upon every subsequent design iteration, the same process is repeated with the 'traditional' approach. Preprocessing the model using the traditional approach was taking between 4 to 8 hours per iteration.

New Approach

Using the new Workbench environment, the same pre-processing task is reduced to around 5 – 10 minutes. This represents a 50-fold productivity improvement. Little to no time is spent on pre-processing which could be considered as a mundane and unproductive activity. The initial impression of the interface was that because of its simplicity, it couldn't be very sophisticated. This first impression was wrong. The user interface is very easy to use and efficient at getting the job done quickly. The automatic mesher is robust and although the user can have control over the mesh parameters, the defaults can be accepted and hence meshing is not one of the required steps. Wizards are available which guide you through the steps involved in an analysis. The process flow diagrams shown below capture the differences in the iterative design process between the traditional and the new approach.

Use of Templates

In every iteration of the design loop, a modified CAD model is read in. The model which is imported need not be a modification of the previous model. It could be a completely different model for a different project. If your products have a set of standard loadcases (eg proof load criteria etc), these can be set up in one analysis and reused for all subsequent analyses by simply importing the relevant geometry into your 'template' analysis and saving the FEA database file under another filename. This improves productivity for the routine analysis jobs.

CAD Integration

The complexity of the geometry of the cast products dictates that the model must be generated with a high end CAD package rather than modeled separately within an analysis package. The CAD model from which production drawings are produced is the same model that is used for stress analysis. The robust and tight CAD / FEA interoperability plays an essential role in gaining productivity improvements.

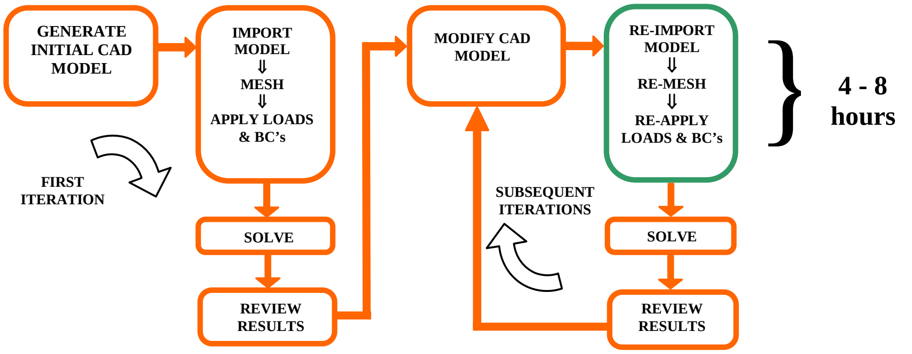
The CAD model and FEA model are associated which can be likened to a spreadsheet embedded and linked to a word processing document. As soon as changes are made to the spreadsheet, these changes are reflected in the word processing document. Likewise, changes made to a CAD model are reflected in the FEA model.

The bi-directional associativity allows two way communication between the CAD package and the FEA package. Namely, ANSYS not only imports the CAD file but it can also control and make modifications to the CAD model. The power of the bi-directional associativity becomes evident when the FEA solver uses the CAD parameters during an optimisation routine. The solver can control the model parameters (eg wall thickness, blend radii etc) to efficiently perform shape optimisation.

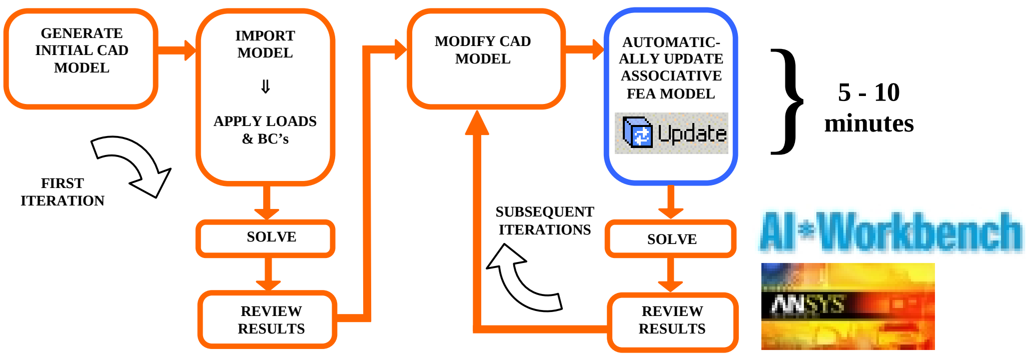
Solution times

Solution times for a typical 165,000 node bogie bolster model are approximately 20 minutes for a linear static solution using a computer with twin 2GHz Intel processors and a Windows based operating system. Typically within half an hour of receiving a CAD model, one can be reviewing and discussing the results. Analysis is no longer a bottle-neck in the design process. The 'what-if' questions are resolved easily and quickly. In a recent example, seven design iterations were completed in a two day timeframe to achieve the desired outcome. Because of the reduced cycle times, the project was not delayed.

Traditional CAD / FEA design process:

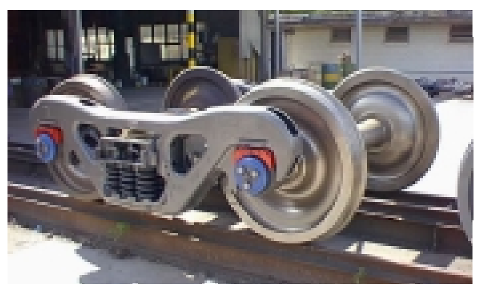


CAD / FEA design process using ANSYS AI*Workbench environment:



Unigraphics CAD model of bogie:

Photograph of bogie:



Finite element analysis of bogie bolster: