Advanced Connecting Rod Analysis Using Custom ABAQUS Plug-Ins

Richard J. Tyrrell, Jesus Blanco

Advanced Analysis Ltd

Abstract: This paper describes the advanced use of ABAQUS and custom built CAE/Viewer plug-ins to analyse connecting rods. The methodology developed by AAL includes machining process effects, generation of oil film elements and a complex load calculation CAE plug-in to represent all gas, inertial and dynamic loads on the connecting rod in a static analysis.

Keywords: Methodology, Plug-ins, Python, Loads Calculation, Connecting rod, Postprocessing, Powertrain, Stress, Visualization.

1. Introduction

General methods for detailed prediction of stresses and fatigue for connecting rods are long established. More often than not, these rely on the use of several separate analysis tools, and their implementation can be cumbersome.

The procedure presented in this paper yields results for every one-degree of crank angle per engine operating condition, while solved in a single static analysis step. Furthermore, all tools required for the complete connecting rod assessment are within the Simulia products domain, with critical usage of CAE/Viewer custom plug-ins. The results from the static analysis compare closely to the equivalent dynamic calculation, with significantly shorter run times and additional benefits.

Three custom built ABAQUS plug-ins were developed by AAL for this methodology:

- The ‘Honing’ plug-in reads deformations from nodes displayed in Viewer. These are used to calculate and include the effects of the honing (machining) process via the command: *IMPERFECTION.
- The 'Oil-Film' CAE plug-in generates an oil film mesh which results in a load distribution which closely represents realistic conditions at small and big end contacts.
- The 'Rod-Loader' CAE plug-in performs complex loading calculations based on external (gas) loads, geometry and materials in the CAE model and dynamic effects. It writes Amplitude tables and a single static Step definition in CAE, which then represents the whole of the engine cycle. It can also be used to represent selected crank angle instants.

2. Overview of Analysis Sequence

Figure 1 shows a breakdown of components in a typical connecting rod analysis.

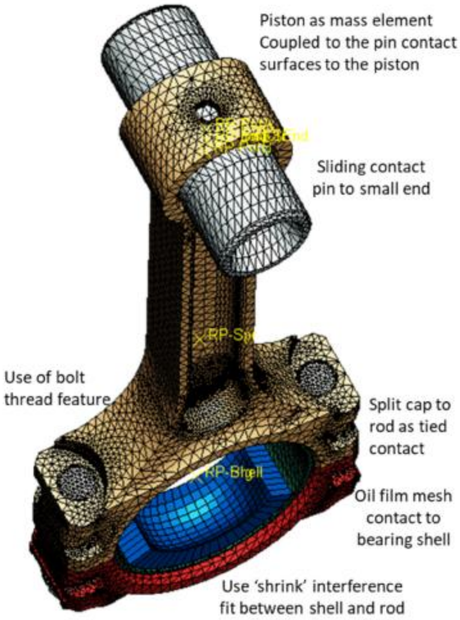

Figure 1. Connecting rod set up in CAE.

The basic process followed for a connecting rod analysis is summarised below. Following sections describe the various plug-ins used in more detail.

- Build a single model in ABAQUS/CAE, including:
  - Connecting rod positioned at TDC non-firing (including any piston pin or crank offset as necessary).
  - Oil film representation, using the 'Oil-Film' plug-in.
  - System to ensure that the small end restraint may be rotated relative to the (nominally stationary) connecting rod, as the rod rotates.
  - Connector element to represent the crankshaft, connected to the centre of the oil film representation.
  - Assembly load step.
  - Running load step, using the 'Rod-Loader' plug-in.
- Export the model with parts turned off ('exclude from simulation') other than the connecting rod and cap, plus the bolts. Suppress all steps except assembly one.
- Perform the honing analysis, which determines the shape of the bore after the honing operation, with bolts tight.

- Use the ‘Honing’ plug-in to modify the bore nodal positions according to the material removed in the honing operation.
- Export the complete CAE model, adding the honing include file.
- Run the main analysis, with two steps: assembly plus operating cycle.

3. ABAQUS Custom Plug-Ins

3.1 Honing Viewer plug-in

Honing is the machining process by which the big end bore of the rod is machined down to a cylindrical shape after bolt assembly loads are applied to the rod. Figure 2 shows idealised versions of the four shapes of a big end bore involved in the complete honing analysis, and after removal of the bolt loads.

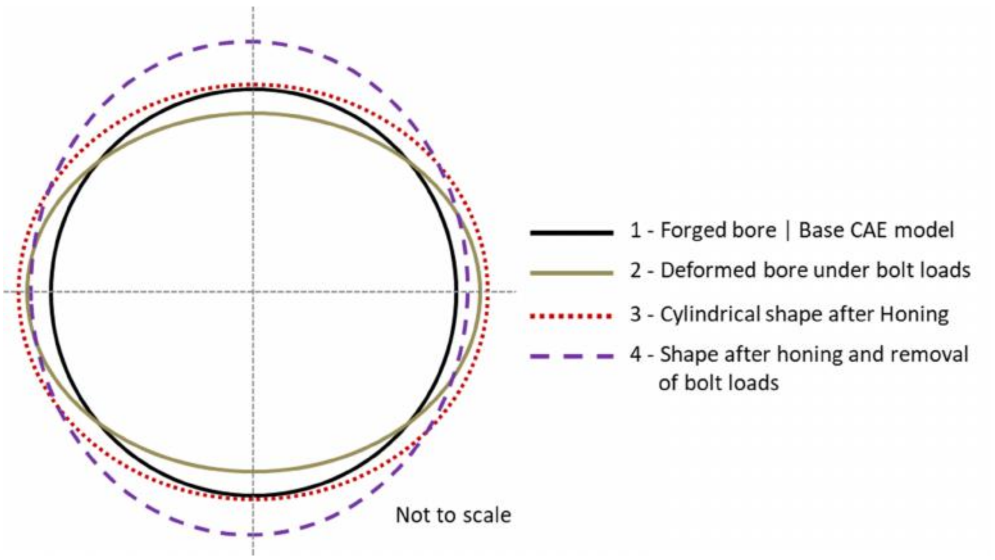

Figure 2. Honing process: big end bore shapes.

Figure 3 shows the GUI for the honing plug-in in ABAQUS/Viewer. The plug-in reads the coordinates and deformations of the bore in Viewer and a user input ‘Honing radius’ value. It then calculates the ‘imperfections’ needed on a nodal basis to simulate this machining process. The plug-in then writes out an ABAQUS include file with the *IMPERFECTION command and nodal entries necessary to move the nodes to the desired locations.

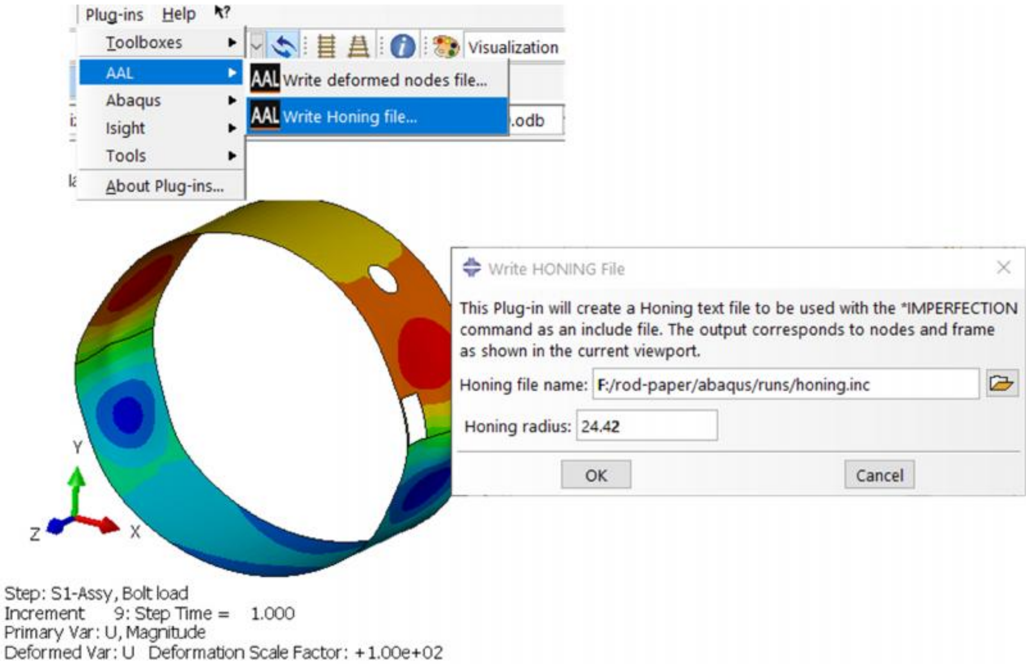

Figure 3. Honing plug-in in Viewer.

Below is an excerpt of the include file generated by the honing plug-in, showing the top and bottom of the file:

```
**
** Output from AAL Plug-in HONING | advancedanalysis.co.uk
*IMPERFECTION, INPUT=F:/rod-paper/abaqus/runs/honing.inc
CAP-1.156,    0.00000,    -0.01054,    0.00000
CAP-1.157,    -0.00407,    -0.01415,    0.00000
CAP-1.158,    -0.00407,    -0.01414,    0.00000
...........
** !! Node below missed honing radius
ROD-1.95735,    0.00000,    0.00000,    0.00000
ROD-1.95811,    -0.01437,    0.00189,    0.00000
ROD-1.96513,    0.01124,    0.00000,    0.00000
**      !! A total of 39 nodes were missed by honing radius
**
**      End of HONING file
**
```

The plug-in also finds the nodes missing the honing radius and writes a comment per each node and the total number of nodes missed at the end of the include file. Therefore, this plug-in is also useful to identify from the analysis if the originally planned bolt load and honing radius will miss some of the material at machining, as the initial model had too large a radius.

3.2 Oil-Film generation CAE plug-in

The oil-film custom plug-in generates a mesh of an oil film mesh which results in a load distribution which closely represents realistic conditions at small and big end contacts of the rod. Further details in “(Tyrrell, 2000)”.
Figure 4 shows the plug-in and the mesh generated in CAE. Furthermore, the plug-in also adds all necessary material, section, sets and surfaces definitions to the CAE database.

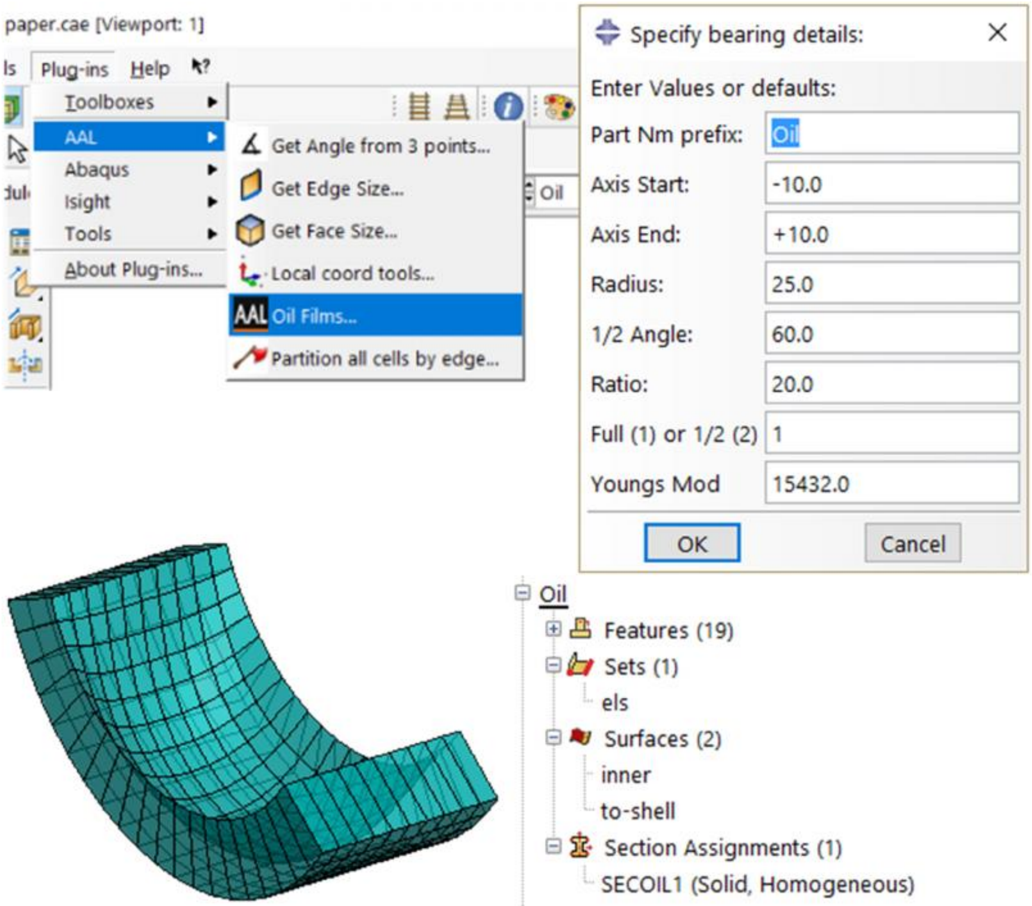

Figure 4. Oil-Film plug-in in CAE.

3.3 Rod-Loader CAE plug-in

The Rod-Loader CAE plug-in performs complex loading calculations based on external (gas) loads, geometry and materials in the CAE model and dynamic effects. It writes Amplitude tables and a single static Step definition in CAE, which then represents the whole of the engine cycle.

Figure 5 shows the plug-in in CAE at the Assembly module. It makes use of the existing geometry to extract design data such as mass, rod length and location of key reference points.

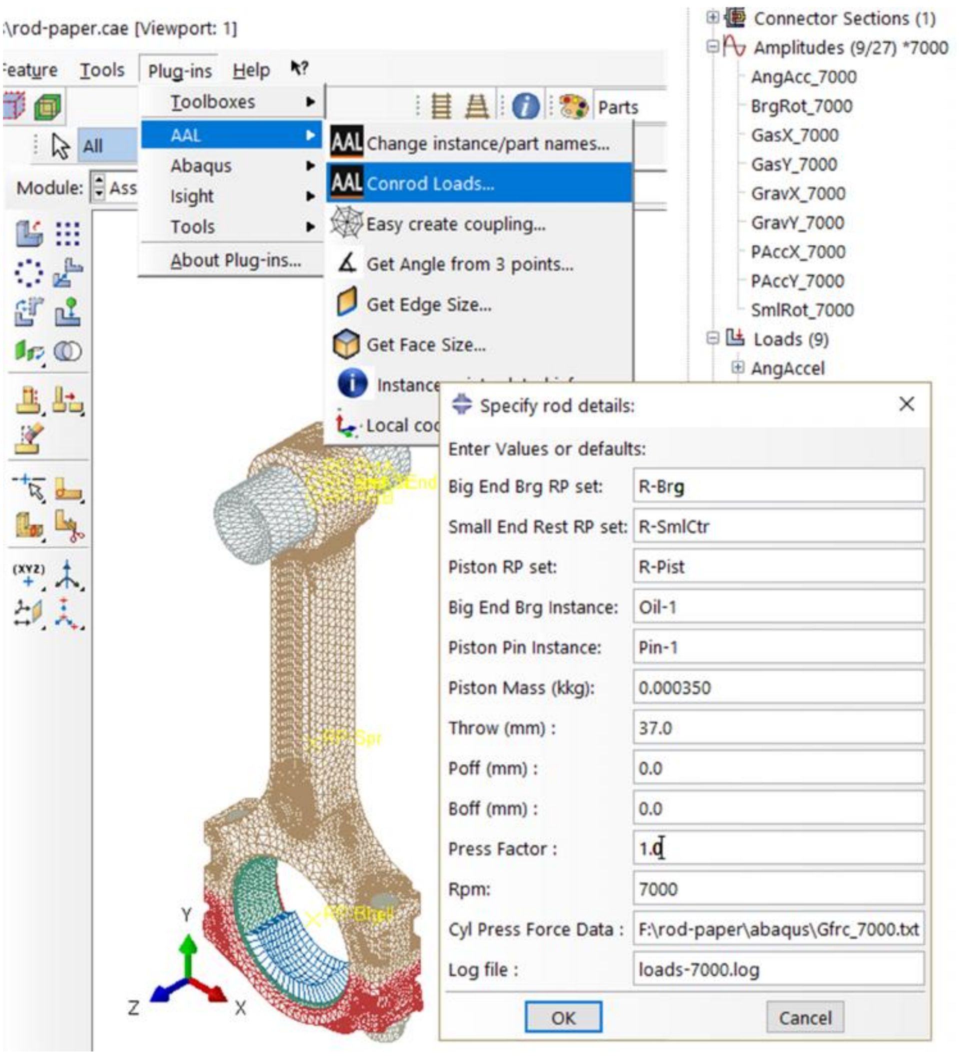

Figure 5. Rod-Loader plug-in in CAE.

In addition to generating all inputs needed in CAE for the analysis, the plug-in generates a comprehensive log file. Figure 6 shows an excerpt of this file with design data and analytical results such as expected forces at big and small ends, accelerations, etc, which are key to perform sanity checks on the results from the analysis.

Model :	Rod-1
Date :	07:51PM on August 21 2017
This log :	F:\rod-paper\abaqus\loads-7000.log
Gas Frc Da	F:\rod-paper\abaqus\Gfrc_7000.txt
Eng RPM :	7000
Throw :	0.037
Rod Lengt	0.146702
Cog2Pist :	0.106072
Big2Cog :	0.04063
Orientatic	Ypos
Piston Ma	0.35
Pin Mass k	0.102916
Rod Assy k	0.461171
Rod Izz kg.	0.001594
CoG Coorc	-0.00818521474796; 40.6299413978; 0.00200880023237

deg	Crrad	Pacc	Gamrad	RotVel	RotAcc	BigAccX	BigAccY
0	0	-24896.2	0	184.882	0	0	-19881.8
1	-0.0175	-24890.3	0.0044	184.855	-2214.9	-347	-19878.7
2	-0.0349	-24872.5	0.0088	184.776	-4429.4	-693.9	-19869.7
3	-0.0524	-24842.8	0.0132	184.644	-6643.4	-1040.5	-19854.5

Selected Loading Instants:

NOTES!

Values here are listed in steadily changing bearing angle.

Values here are those to put into cae. Thus:

- The .inp deck will have Ang Vel as sq of value here
- Grav loads are listed here in m/s2. Multiply by 1000 for mm model

Case	Crank Degs	Small Rot Radians	Brg Rotn Radians	GravX m/s2	GravY m/s2	Ang Acc rad/s2	Ang Vel rad/s
TDCpump	0	0	0	0	21270.55	0	184.88
MinBend	90	-0.25497	-2.28848	13548.69	-5014.44	-140053	0
MaxSide	435	-0.2461	-2.63423	14420.18	411.11	-134363	49.34
MaxComp	375	-0.06532	-3.09941	5045.34	20128.75	-33056.3	178.96
TDCfire	360	0	-3.14159	0	21270.55	0	184.88
BDCpump	180	0	-3.14159	0	-18493	0	-184.88
BDCfire	540	0	-3.14159	0	-18493	0	-184.88
MaxBend	270	0.25497	-3.88761	-13548.7	-5014.44	140052.9	0
MinSide	645	0.2461	-4.6189	-14420.2	411.11	134362.6	49.34
MaxTens	719	0.0044	-6.27641	-344.49	21265.41	2214.85	184.86

Figure 6. Rod-Loader log file output.

The plug-in also outputs at the bottom of the log file selected loading instants that can be used as separate loading conditions for preliminary design optimisation of the rod geometry.

The analytic calculations of forces acting on the connecting rod are well understood, e.g. as described in “(Shenoy, 2004)”.

The ABAQUS results from the static analysis can be post-processed with insightful animations in Viewer without showing rigid body motions of the rod, as loads are calculated and applied to a vertical, nominally stationary rod. The results at every one-degree crank angle allow for consequent detailed fatigue analysis in Fe-Safe. The fatigue calculation is not covered in this paper.

4. Static vs Dynamic Analysis Approach

4.1 Comparison of results Static vs Dynamic

During the development of this methodology extensive checks across results were performed between ABAQUS results for a full implicit dynamic analysis, those obtained in the analytic calculation of loads, and those obtained using the new static analysis. Figure 7 shows a comparison of results for predicted horizontal and vertical loads at the big end.

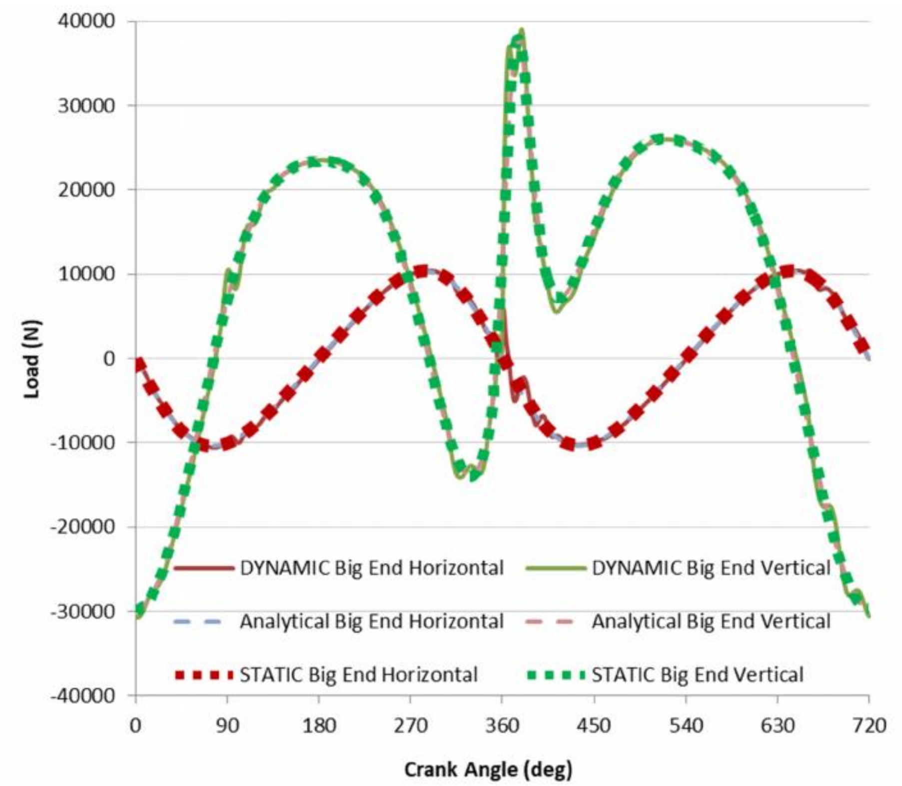

Figure 7. Big End Forces: Analytical vs Static vs Dynamic.

The correlation across different solving methods is very close. Typically the Static analysis produces smoother results than the Dynamic one. Furthermore, the Static run solves at least 2 times faster than the dynamic model. This depends on the design and loads involved as the dynamic analysis may take longer depending on the number of revolutions needed to settle down.

Figure 8 shows a comparison of stress results on the rod at top dead centre (TDC Pumping) for the Static vs Dynamic analyses. The correlation is also very close.

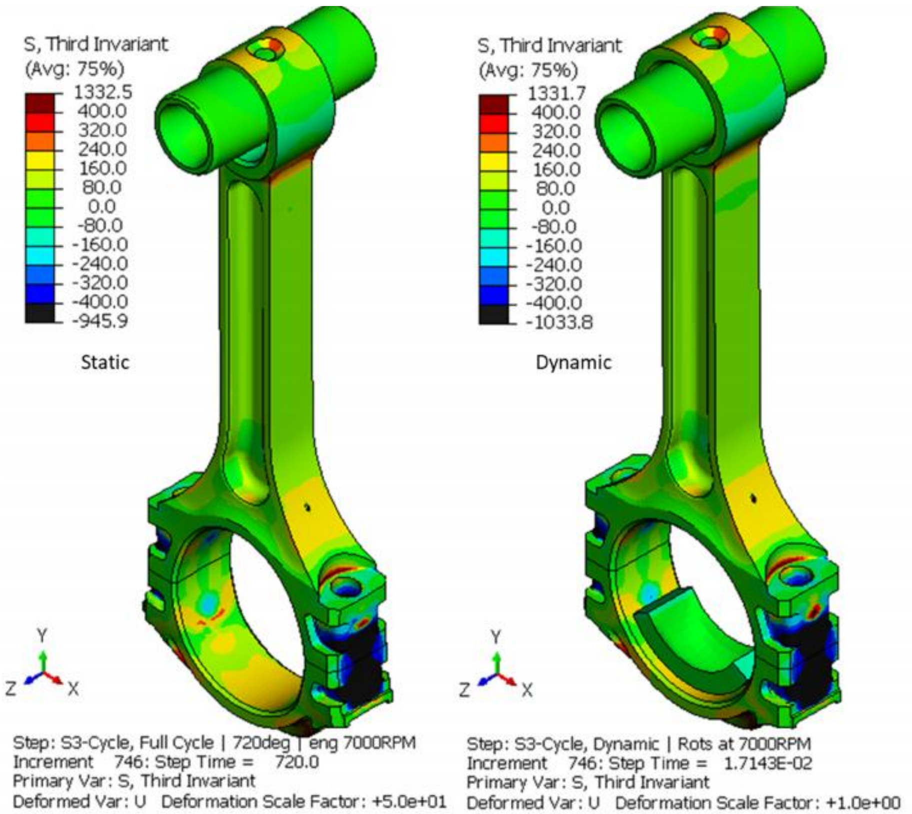

Figure 8. Stress comparison at TDC Pumping: Static vs Dynamic.

The dynamic analysis tends to produce some artificial vibrations, which may make some localised areas to show slight differences when compared to the results from the Static analysis.

Figure 9 shows a comparison of stress results on the rod during peak gas pressure (14ATDC) for the Static vs Dynamic analyses. The correlation is also close.

Note the deformed shapes in Figures 8 and 9. The Static approach allows visualising distortions of the model, while the results from the dynamic analysis are plotted with magnification factor of 1.

The Step Time in the Static analysis solves for a period of 720, which corresponds to the crank angle in degrees. This aids postprocessing when looking at results for specific crank angle instants.

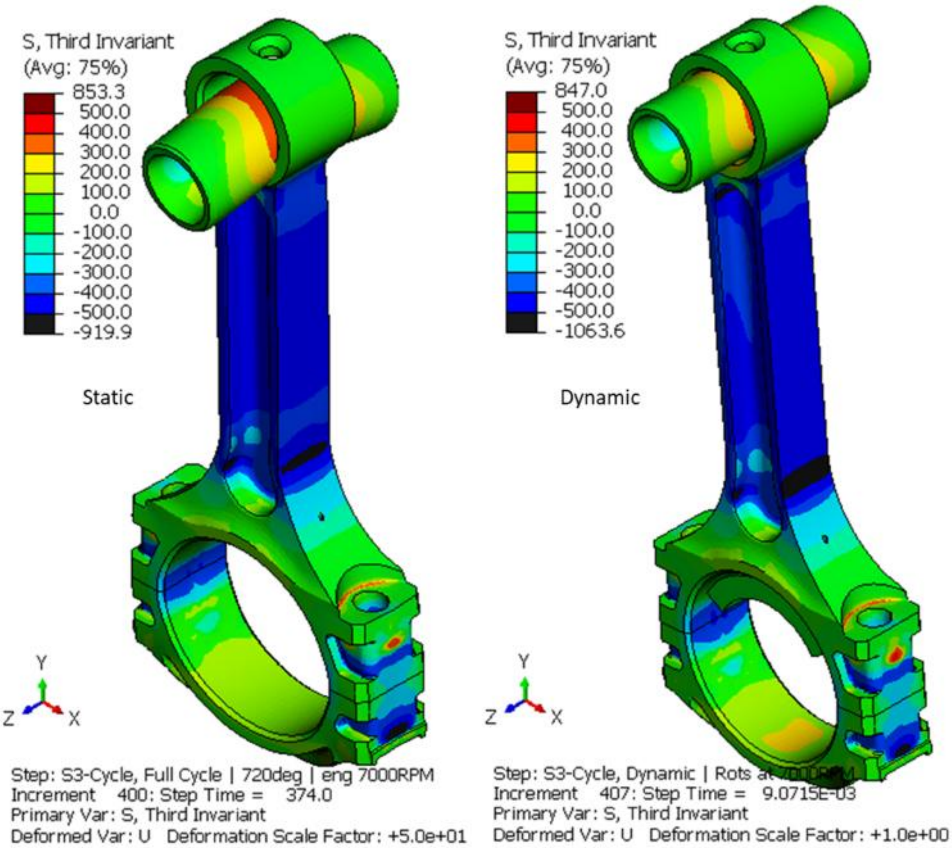

Figure 9. Stress comparison at peak gas pressure: Static vs Dynamic.

4.2 Effects of the oil film on contact pressures

Figure 10 shows contact pressure (CPRESS) distributions on the big end at different crank angles. During peak gas pressure at 14ATDC the parabolic load distribution is clearly shown as the rod undergoes maximum compressive loads.

During TDC as the big end deforms due to large tensile forces the distribution of the load due to the presence of the oil film is quite different than when the rod is under compression. This agrees with “(Sato, 2002)”.

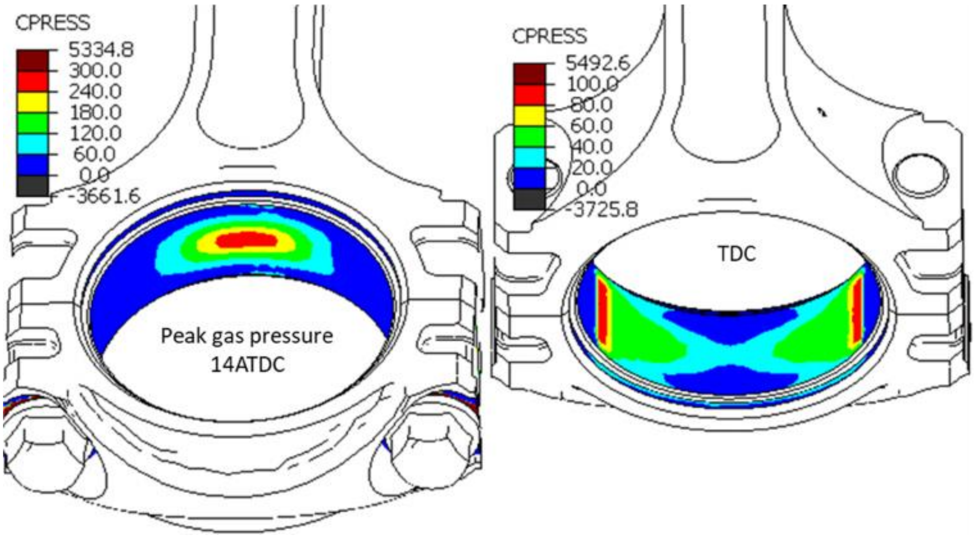

Figure 10. Contact pressure due to oil film at 14ATDC and TDC.

4.3 Benefits of the Static over the Dynamic approach

Following are a number of reasons to support the implementation of this methodology comprising custom built plug-ins and the Static solving approach:

- Static runs are significantly faster than the equivalent Dynamic ones. Typically the speed up rate is about 2 to 3 times depending on the design.
- Results from the Static analysis tend to be smoother, and show closer agreement with analytical results, than with the Dynamic approach.
- The Rod-Loader plug-in allows for selective Static loadcases to be ready for quick checks on a design direction without having to solve the full engine cycle.
- The complete methodology can be implemented in a single ABAQUS environment.

5. Tips for Connecting Rod Analysis

In addition to the features illustrated in Figure 1 the following tips are recommended to improve quality of results and/or speed up analysis convergence:

- In order to analyse a full engine cycle the output of results at each of the 720 degree crank angles means that the results file (.ODB) can be quite large. It is recommended to start with a mesh comprising no more than 100,000 high-order elements for a typical automotive application. In addition, if the number of elements is excessive, run times for the full cycle can increase significantly.

- In the case that after a first run some areas may still need increased mesh density the Sub-modelling technique with small local models should be considered, instead of increasing significantly the size of the global model.
- The bolt thread feature in ABAQUS is also a useful tool to improve the local load paths and stress distribution around the threads under bolt loads without modelling the threads explicitly.
- The split cap to rod should be modelled firstly as a tied contact. This allows for inspection of negative contact pressures (CPRESS) to check for potential separation.
- The addition of a separating contact between the cap and the rod may increase significantly convergence difficulties with the analysis. It is likely that default convergence tolerances may need to be relaxed (*CONTROLS command). Furthermore, the above may not be enough and refinement of the mesh at the contact will be required. This is to reduce the residual force produced at the slave nodes, which may not converge easily when compared to the average force during the non-linear analysis solving.
- Most contact interactions involved in a rod analysis may have friction coefficients between 0.10 and 0.20. Note that any friction coefficient > 0.20 will trigger in ABAQUS the UNSYMMETRIC matrix solver. This means that the analysis will take twice as long to solve. It is recommended to either turn the UNSYM parameter to NO, or reduce friction coefficients to be =< 0.20.

6. Summary

The development of the advanced methodology to analyse connecting rods presented in this paper brings the following main benefits when compared to other approaches:

- Significantly faster solving times with the Static approach compared to a Dynamic analysis.
- The development of the ABAQUS custom plug-ins: ‘Honing’, ‘Oil-Film’ and ‘Rod-Loader’ allow for the addition to the analysis of a complex machining process, realistic pressure distributions with the oil film model, and detailed full engine cycle loads to be applied in a single Static analysis.
- The results from the Static approach tend to be smoother than from an equivalent Dynamic analysis.
- Selected crank angle instants can also be easily implemented during early concept design development.
- The Static approach also allows for insightful deformed shapes of the rod without including rigid body motions.

7. References

1. Sato K., Makino K., Machida K. "A Study of Oil Film Pressure Distribution on Connecting Rods Big Ends," Society of Automotive Engineers, Inc., Detroit, USA, 2002.
2. Shenoy P. S., "Dynamic Load Analysis and Optimization of Connecting Rod," Thesis, University of Toledo, 2004.
3. Tyrrell, R.J. and Concannon, J, "Inclusion of Oil Film Pressure Distribution in Contact Analyses", Abaqus RUM, 2000.