COMPARING CREO SIMULATE WITH CREO SIMULATION LIVE

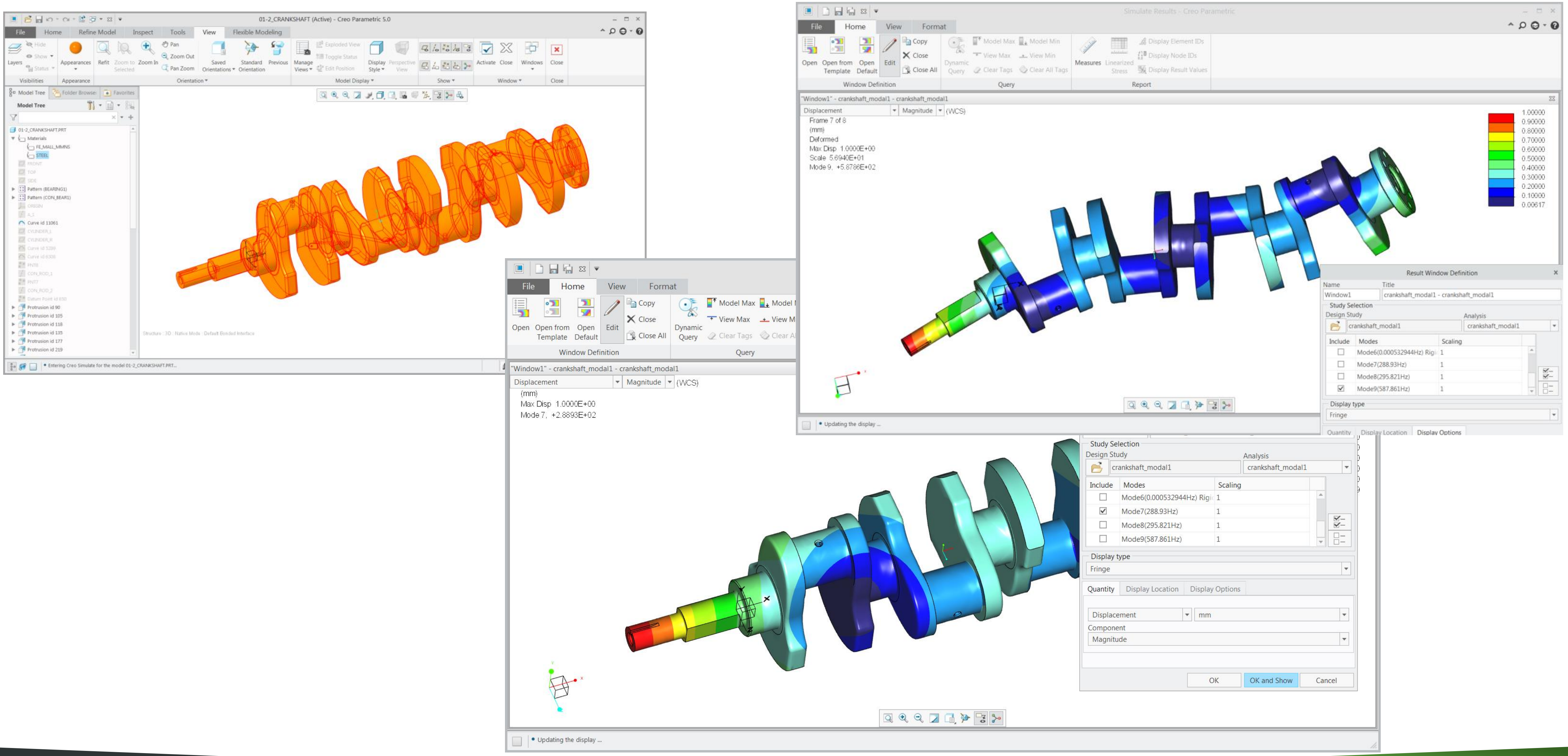
AGENDA

Datasets Used in Comparison

- Engine Crankshaft - Modal Analysis
- V8 Engine – Structural Analysis
- Manifold – Thermal Analysis
- Tuning Fork – Modal Analysis

ENGINE CRANKSHAFT - MODAL ANALYSIS

CREO SIMULATE RESULTS



CREO SIMULATE RESOURCE USAGE

Single Pass Adaptive

3 minutes total time (includes meshing)

```
Memory and Disk Usage:

  Machine Type: Windows 7 64 Service Pack 1
  RAM Allocation for Solver (megabytes): 512.0

  Total Elapsed Time  (seconds): 149.04
  Total CPU Time      (seconds): 186.33
  Maximum Memory Usage (kilobytes): 1525494
  Working Directory Disk Usage (kilobytes): 1207296
```

```
Number of Rigid Modes: 6

Number of Modes: 9

   Mode   Frequency (Hz)
   ----   --------------
      1    0.000000e+00
      2    0.000000e+00
      3    0.000000e+00
      4    0.000000e+00
      5    0.000000e+00
      6    5.329435e-04
      7    2.889300e+02
      8    2.958206e+02
      9    5.878609e+02
```

```
RMS Stress Error Estimates:

Mode    Stress Error  (% of Max Modal Stress)
----   --------------
   1        2.7%
   2        2.1%
   3        2.5%
   4        3.6%
   5        2.6%
   6        2.1%
   7        2.6%
   8        3.4%
   9        2.7%
```

CREO SIMULATION LIVE RESULTS 293.3 / 595.6

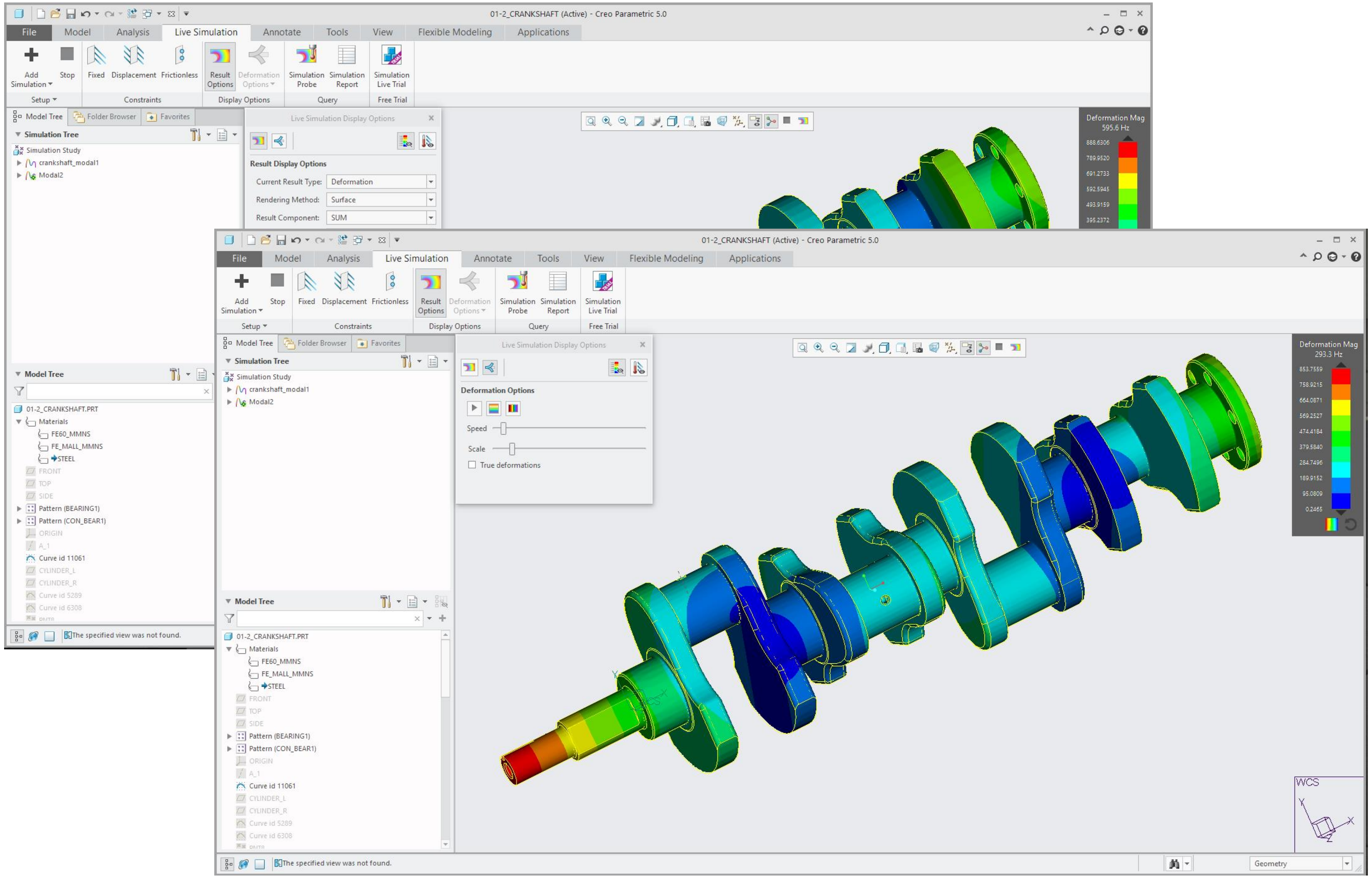
Results

Mode 1: 293.3Hz

Mode 3: 595.6Hz

Time taken

< 2 seconds



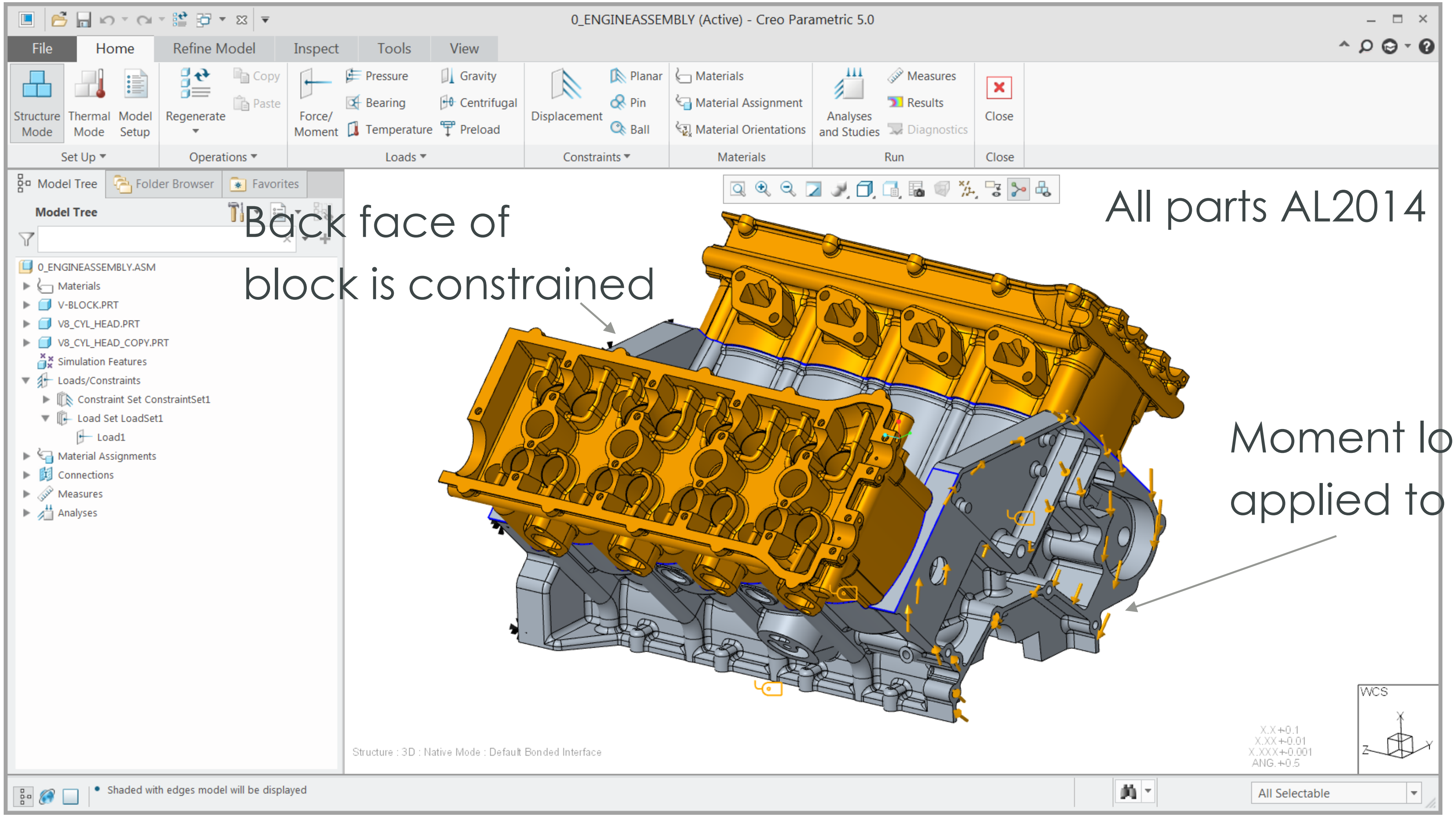
RESULTS COMPARISON

ptc

	Creo Simulation Live	Creo Simulate	% difference
Mode 1	293.3 Hz	288.9 Hz	1.5%
Mode 3	595.6 Hz	587.9 Hz	1.3%
Solution Time	3 sec	3 minutes	60x

V8 ENGINE - TORQUE LOAD

V8 ENGINE MODEL – STATIC ANALYSIS



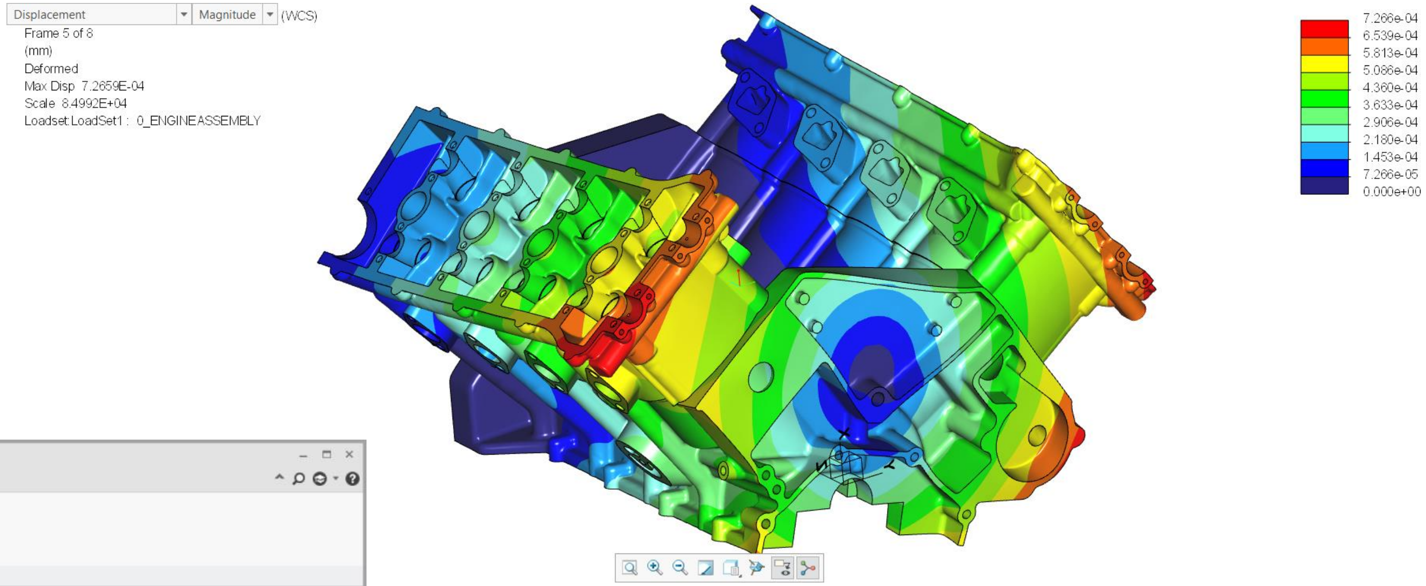
Back face of block is constrained

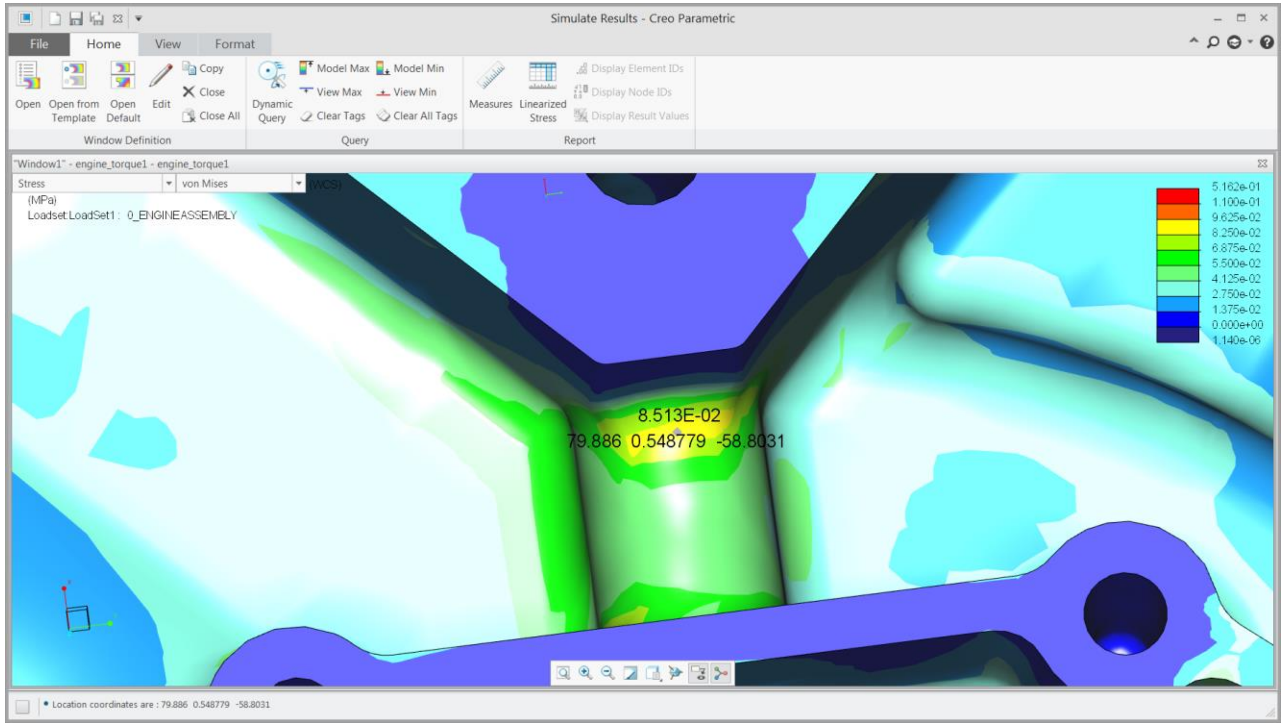
All parts AL2014

Moment load applied to face

CREO SIMULATE RESULTS

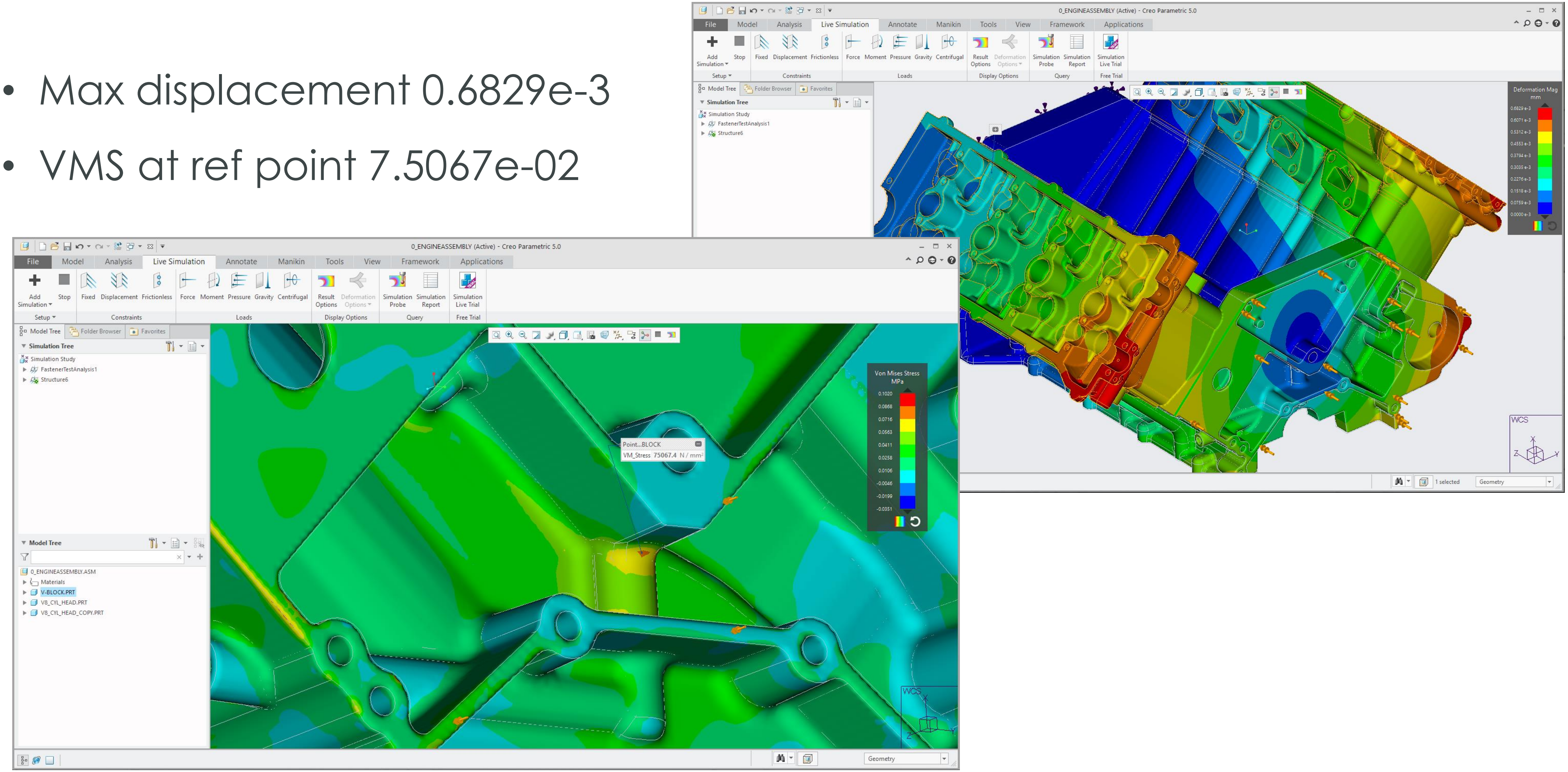
- Max displacement 0.7266e-3
- VMS at Ref point 8.513e-2





CREO SIMULATION LIVE RESULTS

- Max displacement 0.6829e-3
- VMS at ref point 7.5067e-02



RESULTS COMPARISON

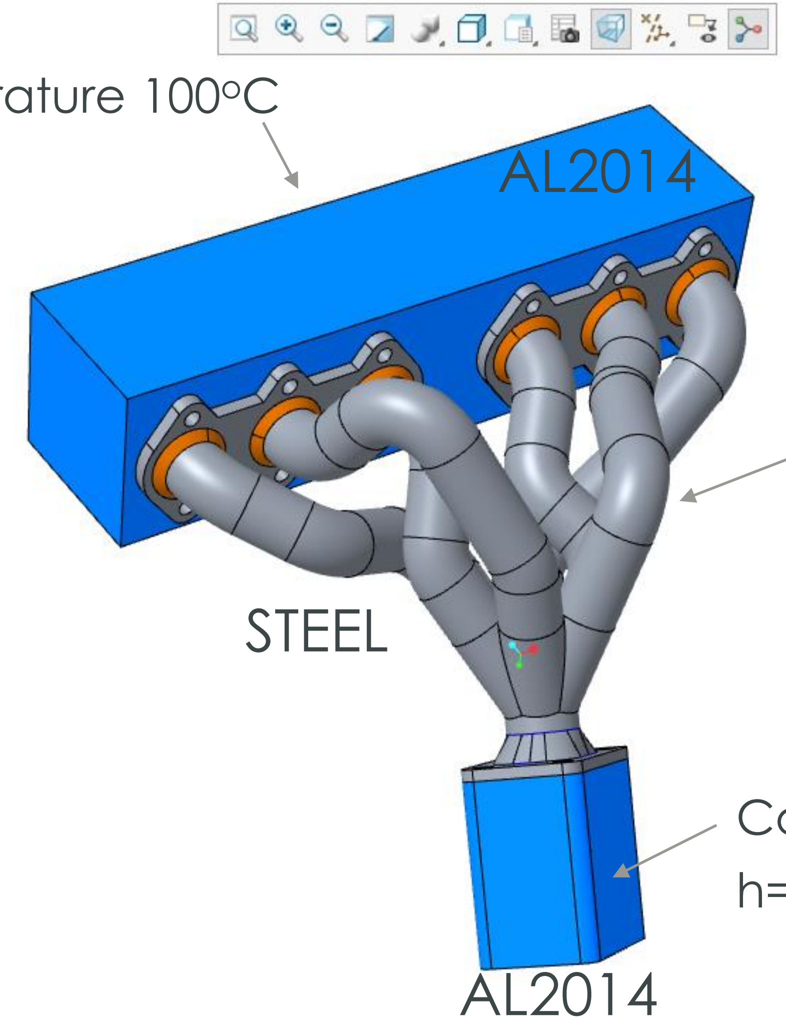
ptc

	Creo Simulation Live	Creo Simulate	% difference
max displacement	0.6829e-3 mm	0.7266e-3 mm	6%
Stress at Ref point	7.5067e-02	8.513e-2	12%
Time for single solution	<10 sec	3 hours	!!!!! ☺

MANIFOLD – THERMAL ANALYSIS

STEADY STATE THERMAL MODEL

Prescribed temperature 100°C



Convection

Inside h=0.5 mW/mm^2K, T-air=800°C

Outside h=0.1 mW/mm^2K, T_air=150°C

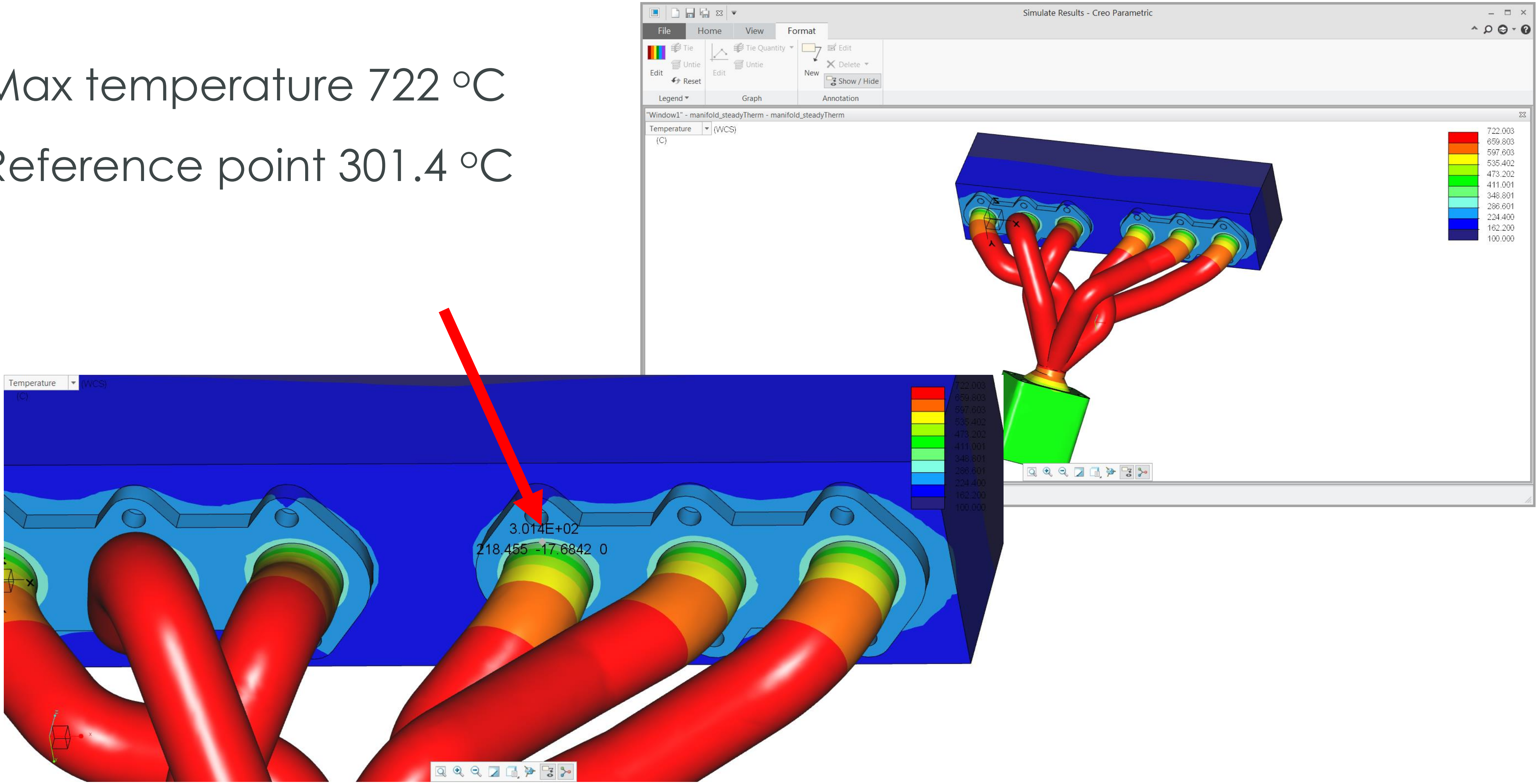
Convection

h=0.02mW/mm^2K, T_air=150°C

X.X+-0.1
X.XX+-0.01
X.XXX+-0.001
ANG.+-0.5

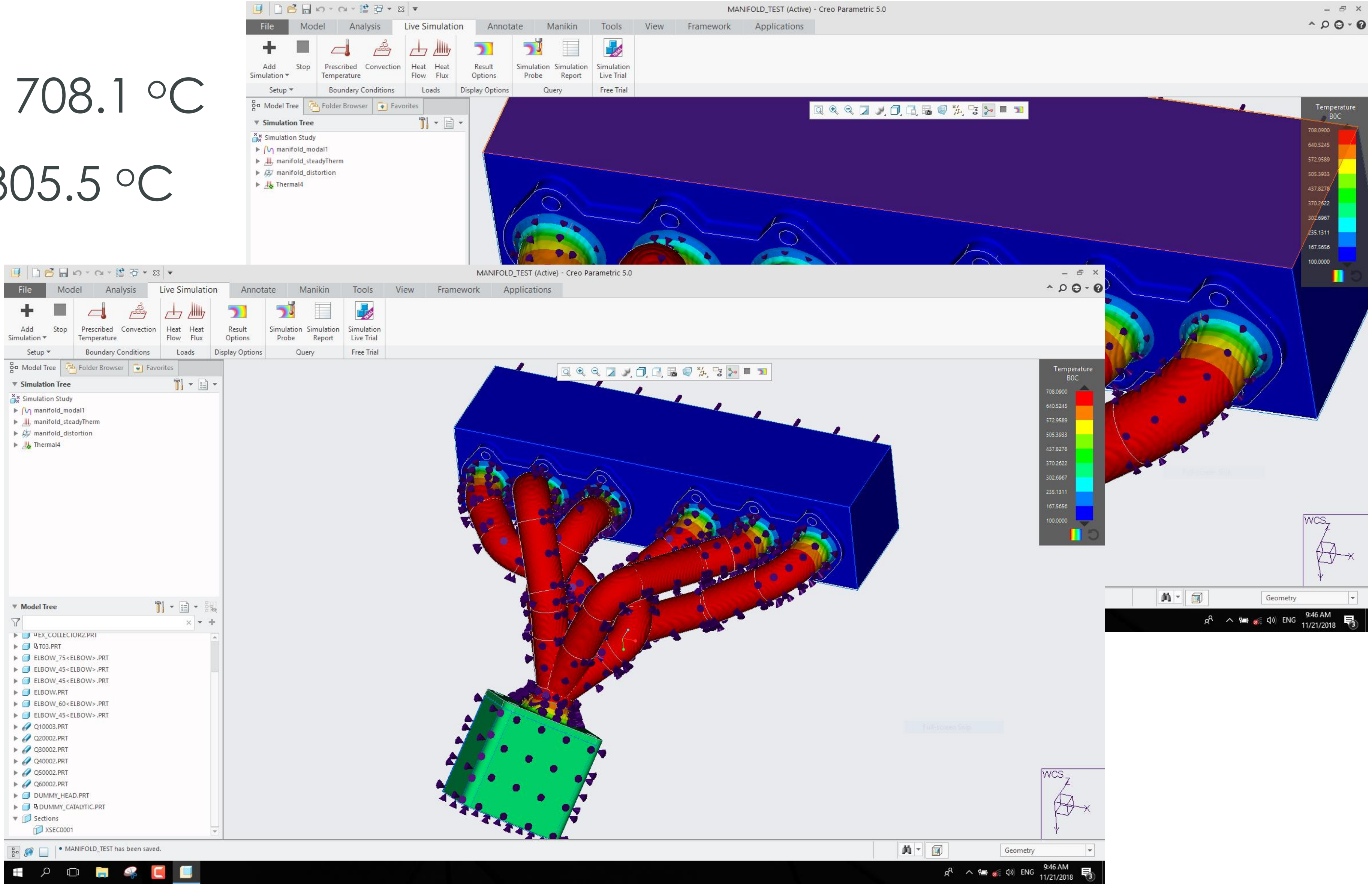
SIMULATE RESULTS

- Max temperature 722 °C
- Reference point 301.4 °C



CREO SIMULATION LIVE RESULTS

- Max temperature 708.1 °C
- Reference point 305.5 °C



RESULTS COMPARISON

ptc

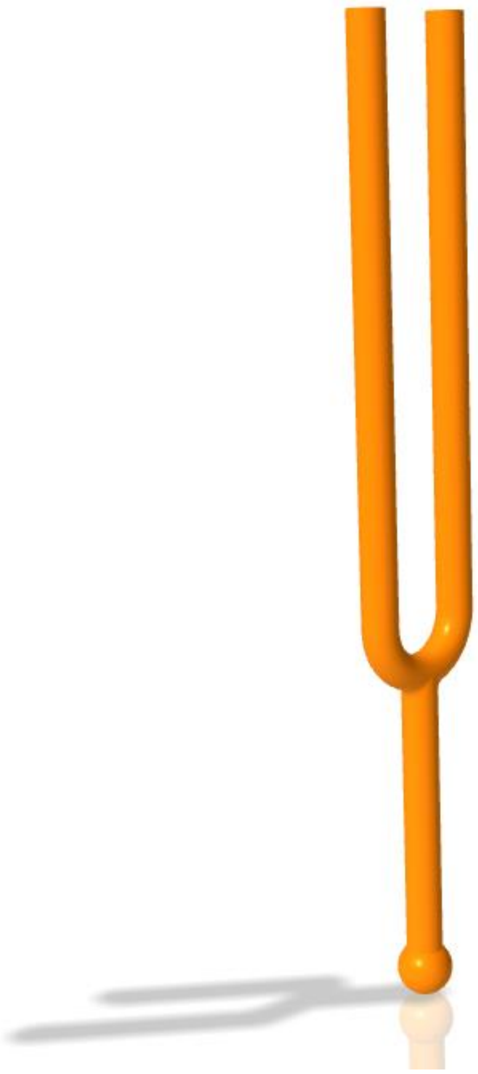
	Creo Simulation Live	Creo Simulate	% difference
Max temperature	708.1 °C	722.0 °C	1.9%
Ref point temp.	305.5 °C	301.4 °C	1.3%
Solution Time	<1 sec	1 minute	60x

* Steady State thermal analysis is AMAZING fast

TUNING FORK – MODAL ANALYSIS

TUNING FORK ANALYSIS

- Used for tuning musical instruments
- A standard reference frequency is 440Hz (A above middle C)
- This frequency is produced when the two "legs" are oscillating
- The first model of vibration is the dominant one, and the thus the frequency of the first mode should be 440Hz

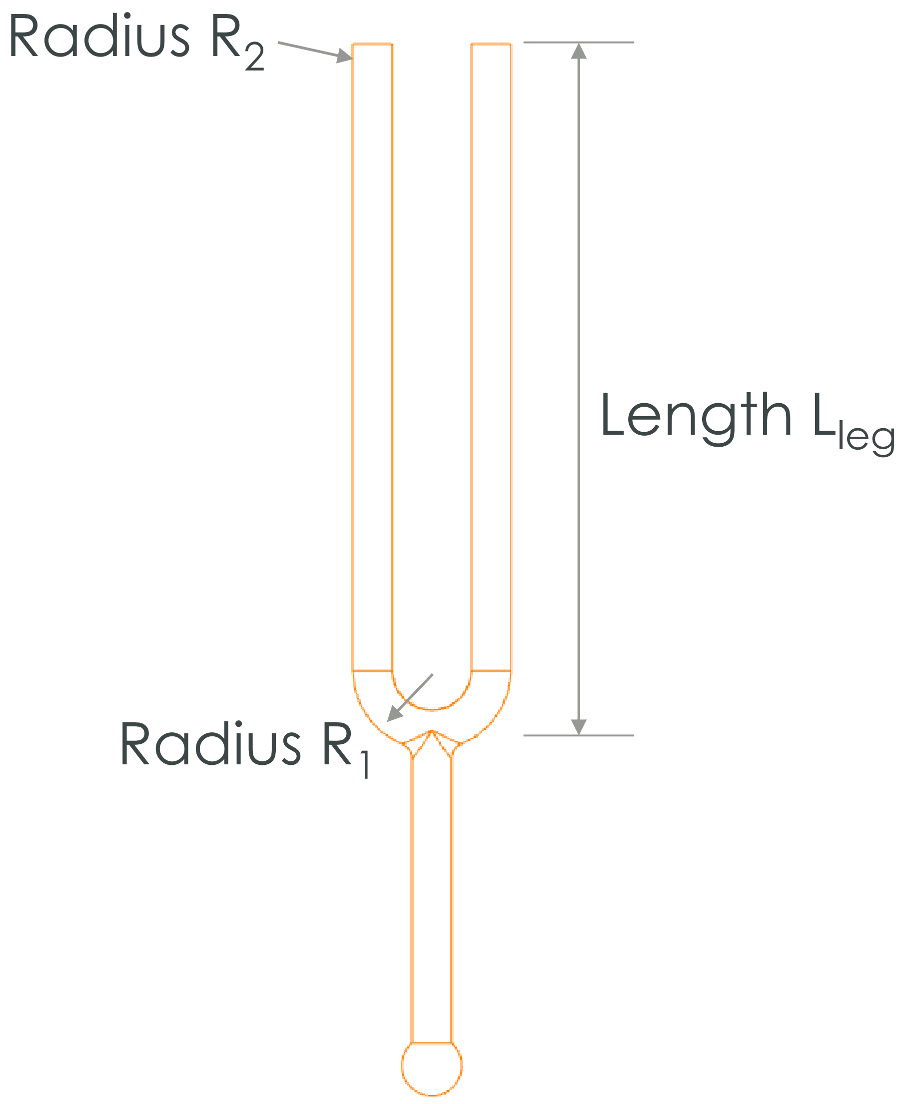


THEORY

- In a rigidly constrained cantilever beam the theoretical* natural frequency is

$$f = \frac{1.875^2 . R_2}{4\pi L_{leg}{}^2} . \sqrt{\frac{E}{\rho}} \qquad (1)$$

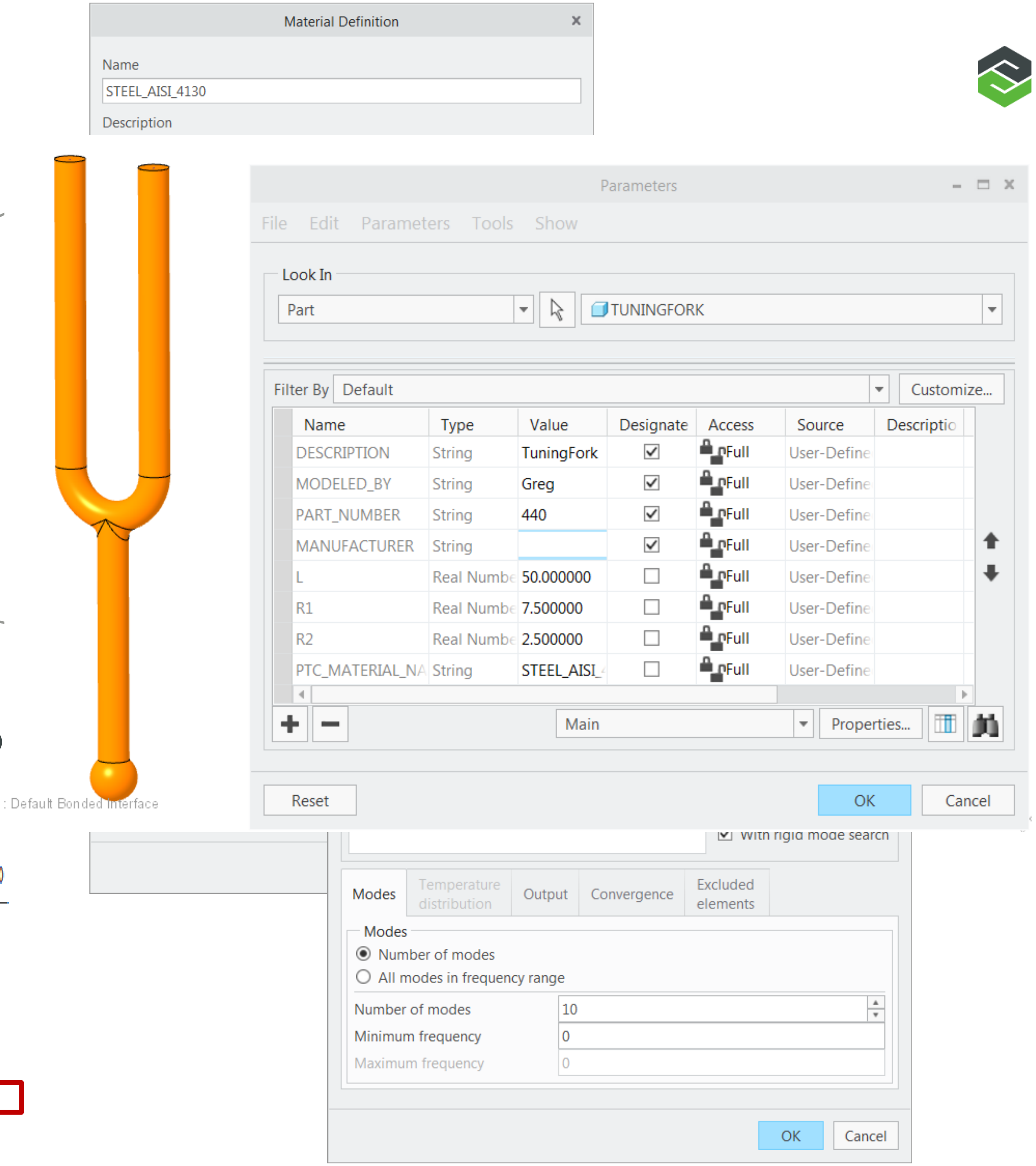
- Where $L_{leg} = L + \frac{1}{2}\pi R_1$
- Material: AISI4130 where
  E = 205GPa, and ρ = 7850 kg/m3
- R1=7.5mm, R2=2.5mm
- For f=440Hz, → L=78mm
- This will be slightly underestimated due to slight differences due to cantilever beam approximation of real geometry



CREO SIMULATE MODEL

- Geometry as shown
  - Set parameter L =50mm
- Material assigned
- Create Modal analysis
  - Unconstrained
- Results
  - Look for first non-rigid mode (should b the two fork legs moving symmetrically)
  - So for L=50 → f=972Hz

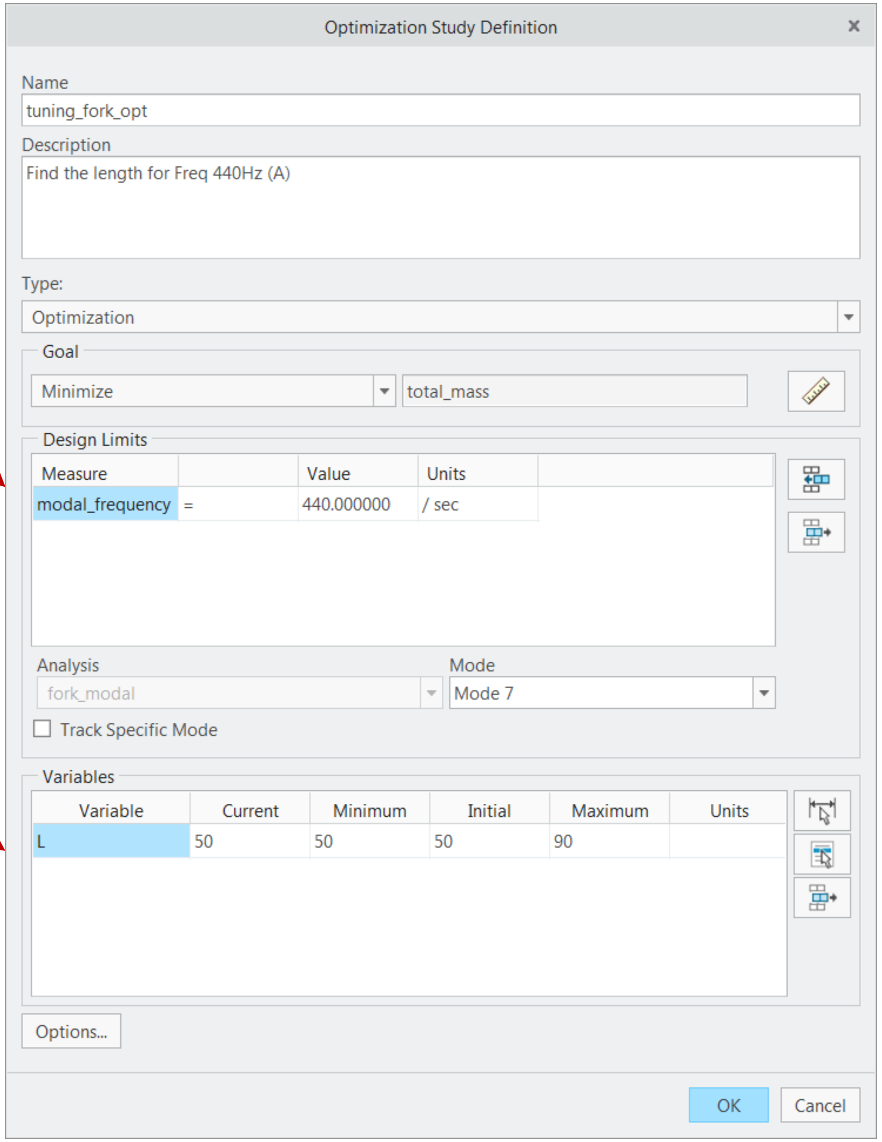
Mode	Frequency (Hz)
1	0.000000e+00
2	0.000000e+00
3	0.000000e+00
4	0.000000e+00
5	2.314839e-03
6	4.082460e-03
7	9.718596e+02
8	1.474661e+03
9	1.881424e+03
10	1.926127e+03



OPTIMIZING THE MODEL

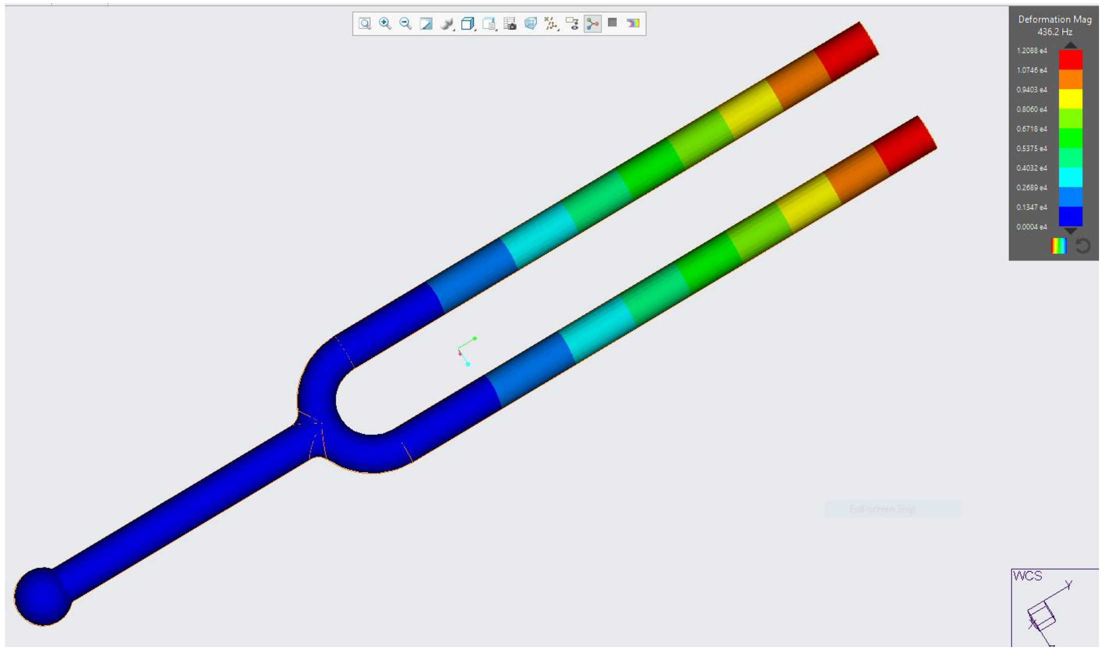
- We want to determine length L to give **f=440Hz**
- Use **Optimization Study** in Creo Simulate
- Set the Design Limit to be 440Hz on Mode 7
- Assign parameter L some upper/lower limits
- Run the study
  - Result… **L=79.4mm (giving f=440.4Hz)**

```
    Parameters:
       L                       79.3952
    Status of Optimization Limit: 1
    modal_frequency    4.4035e+02 =   4.4000e+02 (satisfied
within tolerance)
```



COMPARING TO CREO SIMULATION LIVE

- The geometry/material from previous analysis used directly
- Unconstrained modal analysis defined
- Results: f=436.2 Hz
- This is **about a 1% difference** from Creo Simulate ☺



FINAL COMMENTS

- The technology used in Creo Simulation Live is shown in these examples (and by ANSYS benchmarks) can clearly be seen to be accurate.
- As long as the physics of the problem can be modeled with linear static, modal and steady state thermal assumptions, then the results in Creo Simulation Live will not differ by any significant amount from traditional FEA codes such as Creo Simulate, and many others.
- Note: In very large models such as the V8 Engine example, there is more chance that small geometric features may be approximated/ignored, thus leading to larger differences in stress results. For the purposes of providing design guidance or evaluating typical design level what-if questions, these differences will have little influence. (compare to other uncertainties in material properties, load values, etc.)
- The value of having near real-time results feedback, completely inside the design environment is enormous, and will revolutionize the way simulation is used to drive designs!

ptc