WHITE PAPER

Computer-Aided Engineering
ANSYS® Mechanical™ on Intel® Xeon® Processors

Engineer Productivity Boosted by Higher-Core CPUs

Engineers can be significantly more productive when ANSYS® Mechanical™ runs on CPUs with a high core count.

Executive Summary

In an effort to save costs, some engineers run ANSYS® Mechanical™ on systems with a small number of cores or select an ANSYS® licensing option that restricts ANSYS processing to only two cores. At the same time, the CPU core counts of Intel® Xeon® processors continue to increase in new releases, and the multicore efficiency of ANSYS Mechanical has also continued to advance. These developments now amount to dramatic performance improvements in ANSYS Mechanical, making systems with higher core counts—and a licensing option that takes advantage of those cores—an especially compelling choice that can result in enormous savings in the long run.

Benchmark tests performed on newer systems running ANSYS Mechanical show that Intel Xeon processors using higher core counts can analyze modeled structures many times faster than CPUs using fewer cores can. For benchmark tests requiring nonlinear analysis, a system with 32 cores enabled performed at an average rate that was nearly seven times faster than a system with only two cores enabled.[1] This time saved can amount to up to 80 hours or more of productivity gains per year, per user. Such productivity gains can translate to significant financial and strategic opportunities—through increased engineering resources and faster releases of engineered products—that are ultimately more important to a business' profitability than any short-term savings resulting from reduced hardware and software investments.

Structural Analysis in ANSYS Mechanical

Computer-aided engineering (CAE) allows companies to use simulations to design and stress-test products without physical prototypes. ANSYS is a world leader in such CAE software. Its product, ANSYS Mechanical, is a finite element analysis (FEA) tool that allows designers and engineers to perform structural analysis of physical systems through a software simulation. The FEA calculations involved can take hours or days to perform. Engineers are therefore often looking for high-performance workstations or clusters that allow them to complete their workloads as efficiently as possible.

How Do CPU Core Counts Impact Engineer Productivity?

Mechanical engineers and IT decision makers who are considering the purchase of new systems for FEA work need to know how various system components can affect productivity. This knowledge can help them determine which specifications they require for engineering workstations and clusters.

When considering CPU specifications for new systems based on Intel Xeon processors, some might question whether increasing the number of cores available to ANSYS Mechanical significantly impacts the speed with which engineers can perform structural analysis. After all, not all computing tasks are made faster by a higher core count—an application can benefit from multiple cores only if its underlying code is written to take advantage of multiple cores.

To be clear, the performance of ANSYS Mechanical does indeed benefit from multiple cores. But even so, for those determining the best CPU specifications for their engineering systems, an important question remains: *to what degree* does increasing the available CPU core count of a CAE system accelerate task execution in ANSYS Mechanical?

This paper looks to answer this specific question about core counts by presenting the results of five ANSYS benchmark tests that measured performance within representative workflows.

An Overview of the FEA Workflow

Before we look at the five benchmark test results, it can be helpful to review the workflow that engineers use to perform FEA and to see where the jobs performed in these benchmark tests fit within the overall FEA process. The workflow consists of the three following stages, all of which an FEA engineer completes in ANSYS Mechanical:

1. The first stage of the FEA workflow is *pre-processing*. In this stage, the engineer begins by importing a model that was designed by using a solid modeling computer-aided design (CAD) application. The engineer then creates a mesh for the model. The mesh is a close approximation of the original imported model that divides the solid into many polyhedrons. An example mesh is shown in Figure 1. The pre-processing phase, and the creation of the mesh, is interactive and memory-intensive. It can also be CPU-intensive, especially for automated parametric workflows. The fact that this stage is interactive means that the speed with which this stage is completed depends in large part on the speed of the engineer in addition to the speed of the system.

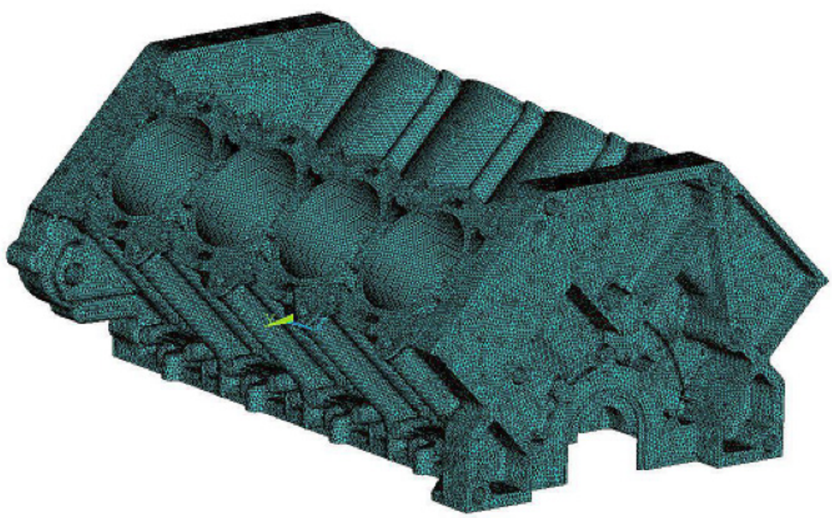

Figure 1. An example mesh used in FEA

2. The second stage of the FEA workflow is the *solution*. In this stage, the analyst tests the simulated mesh structure by placing a stress on it. To begin this step, the analyst enters boundary conditions that, for example, displace a particular node or point on the mesh. The analyst also provides information about the stiffness of the source material. Next, the analyst clicks **Solve** in ANSYS Mechanical, which sets off a long series of calculations in a batch process. This set of calculations can take seconds, minutes, hours, or even days to complete, depending on the complexity of the simulation and the resources available in the system. When the entire problem is solved, ANSYS Mechanical displays the model with detailed information about how the original stress placed on the model has affected nodes or points throughout the structure.

   The solution stage of the FEA workflow is not interactive after the boundary conditions are entered, which means that the completion time of this stage relies almost completely on the speed of the system, not the engineer. The long batch process is memory-intensive, memory-bandwidth-intensive, and CPU-intensive. This is also the workflow stage for which ANSYS makes benchmark tests available, including the tests presented in this paper.

3. The final stage of the FEA workflow is *post-processing*. In this stage, the engineer analyzes the quality of the solution, determines the most important results from the solution stage (such as whether a previous stage needs to be performed again with modifications), and eventually extracts results to present to others, if necessary.

   The post-processing stage is interactive. It is memory-intensive and GPU-intensive, but it is less CPU-intensive than the first two stages.

This breakdown of the FEA workflow shows that the solution stage is the most highly variable in terms of its completion time. Furthermore, because this stage is not interactive like the other phases, it is also the most dependent on system resources as opposed to the speed of the engineer. And because the time required to solve the model can last up to hours or days, especially on a low-powered system, it is the most important stage for which to gauge completion times and to run benchmark tests.

System Specifications

To measure how completion times in the solution stage of the FEA workflow are affected by core counts, an ANSYS engineer performed the five benchmark tests several times on a single system, changing only the number of cores enabled each time.

The engineering workstation used for the benchmark testing was fitted with the following components:

- **CPU**: Two 22-core Intel Xeon processor E5-2699 v4
- **Memory**: 64 GB RAM
- **Storage**: Two Intel® Solid-State Drive (SSD) 750 Series Peripheral Component Interconnect Express* (PCIe*) drives configured in RAID 0

New Features and Improvements in the Parallel Processing Capabilities of ANSYS® Mechanical™

ANSYS Mechanical has been updated in the following ways to take advantage of parallel processing on multiple cores:

New Parallel Features:

- New domain-decomposition algorithms for harmonic analyses and cyclic-symmetry modal analyses
- Restarts with changing core counts
- Support for mesh nonlinear adaptivity

Improved Parallel Performance and Scaling:

- Improved performance for Sparse solver and the Block Lanczos algorithm
- Improved scaling when CE/CPs exist across domain boundaries
- Improved performance for the QRDAMP method with many modes

To begin, the ANSYS engineer ran the first benchmark test with only two cores enabled, as a way to emulate a low-cost ANSYS licensing option (which restricts processing to two cores). He then re-ran the first test with 8, 16, and 32 cores enabled, corresponding to other common ANSYS licensing levels. Finally, he repeated the same process for the remaining tests.

Benchmark Test 1 of 5: Peltier Cooling Block (or “SP-1”)

The ANSYS SP-1 benchmark test places a stress on a thermoelectric-cooling-block model. The analysis performed to solve the test is nonlinear, which means that it accounts for changes in the stiffness of the material as stress is placed on it. Nonlinear analyses are especially CPU-intensive because they naturally require more computations as they account for these changes.

Table 1 provides basic information about benchmark test SP-1, and Figure 2 shows the mesh used for it.

Table 1. Information about the ANSYS® Peltier Cooling Block (SP-1) benchmark test for ANSYS® Mechanical™

Analysis Type	Static Nonlinear Thermal-Electric Coupled Field
Number of Degrees of Freedom	650,000
Equation Solver	Sparse
Matrix	Non-symmetric (Unsymmetric)

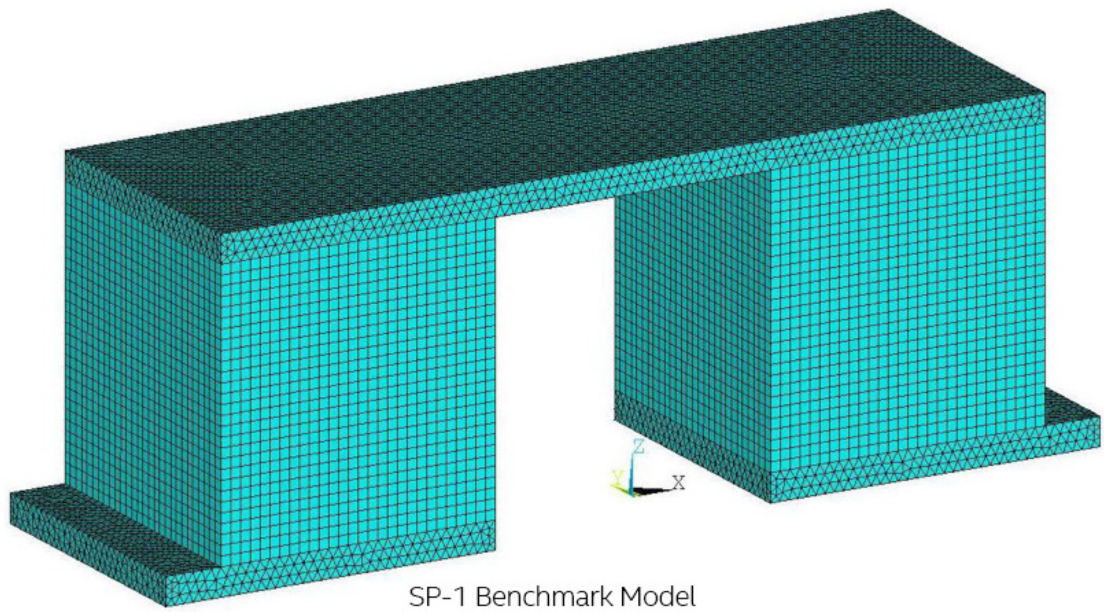

Figure 2. The mesh of the cooling block used in benchmark test SP-1

Benchmark Test 1 (SP-1) Results

For the baseline of two cores, the Intel Xeon processor–based system completed the benchmark test in 17 minutes and 9 seconds. Table 2 shows the time required to solve benchmark test SP-1 with more cores enabled. The time to solve the model decreased with each higher core count, down to a minimum of 2 minutes 23 seconds with 32 cores enabled on the system.[1]

Table 2. Results of benchmark test SP-1[1]

Cores enabled	Solution time
2	17 minutes, 9 seconds
8	5 minutes, 41 seconds
16	3 minutes, 29 seconds
32	2 minutes, 23 seconds

Figure 3 shows the performance improvements of the higher-core systems. On the left, the solution time for the two-core system is represented with a baseline value of 1; the bars to the right show how many times faster the benchmark test was performed with the higher core counts. At this rate of performance scalability, a complex analysis that might require 7 hours to solve on a two-core system would require less than 1 hour to solve on a 32-core system.

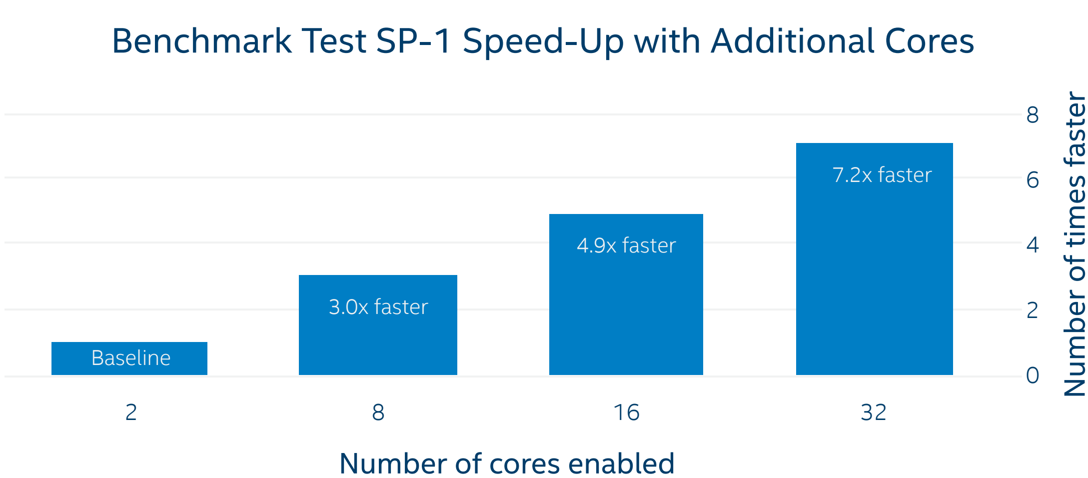

Figure 3. Performance improvements for benchmark test SP-1 as more cores were added[1]

A Note on ANSYS® Benchmark Tests

ANSYS makes available a suite of benchmark tests that can be used to compare the performance of different hardware platforms when running ANSYS solvers. For more information about ANSYS® Mechanical™ benchmarks, visit **ansys.com/solutions/solutions-by-role/it-professionals/platform-support/benchmarks-overview/ansys-mechanical-benchmarks**.

Benchmark Test 2 of 5: Semi-Submersible (or "SP-2")

The ANSYS SP-2 benchmark test places a stress on a model of a semi-submersible drilling rig. As with the first benchmark test, the analysis performed in this test is nonlinear and therefore especially CPU-intensive.

Table 3 presents basic information about this benchmark test. Figure 4 shows both the original solid model (left) and the mesh (right) on which the test was performed.

Table 3. Information about benchmark test SP-2 for ANSYS® Mechanical™

Analysis Type	Transient Nonlinear Structural
Number of Degrees of Freedom	4,700,000
Equation Solver	Sparse
Matrix	Symmetric

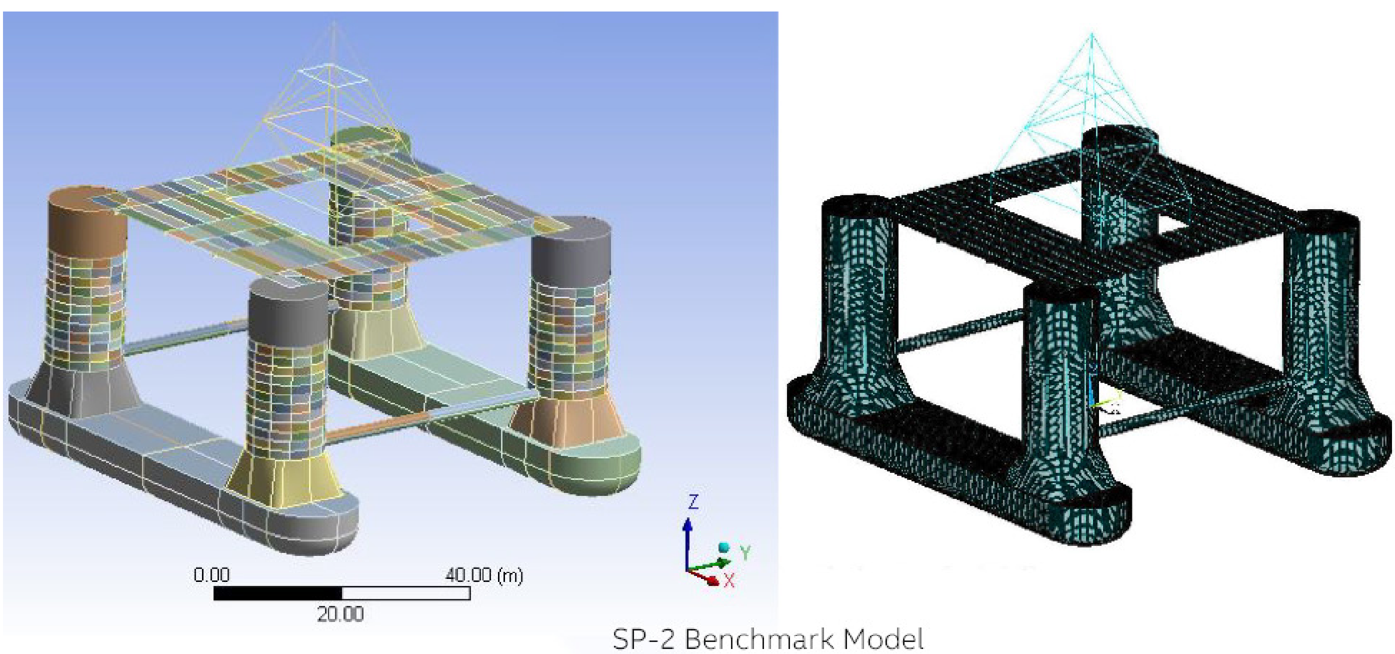

Figure 4. The SP-2 benchmark model before and after the meshing process

Benchmark Test 2 (SP-2) Results

With two cores enabled, the Intel Xeon processor–based system completed the SP-2 benchmark test in 38 minutes and 49 seconds. Table 4 shows the complete test results.[1]

Table 4. Results of benchmark test SP-2[1]

Cores enabled	Solution time
2	38 minutes, 49 seconds
8	12 minutes, 9 seconds
16	7 minutes, 59 seconds
32	5 minutes, 48 seconds

Figure 5 presents the performance scalability associated with higher core counts. As with Figure 3, the time required to complete the benchmark test for a two-core system is represented on the left in Figure 5 with a baseline value of 1. The bars on the right show how many times faster the same benchmark test was completed with the higher core counts.

Figure 5 shows that the performance-improvement ratios for benchmark test SP-2 are similar to those seen in the first benchmark test. At this rate of performance scalability, a solution that takes 7 hours to complete on a system with two cores enabled would take a little over an hour to complete on a system with 32 cores enabled.

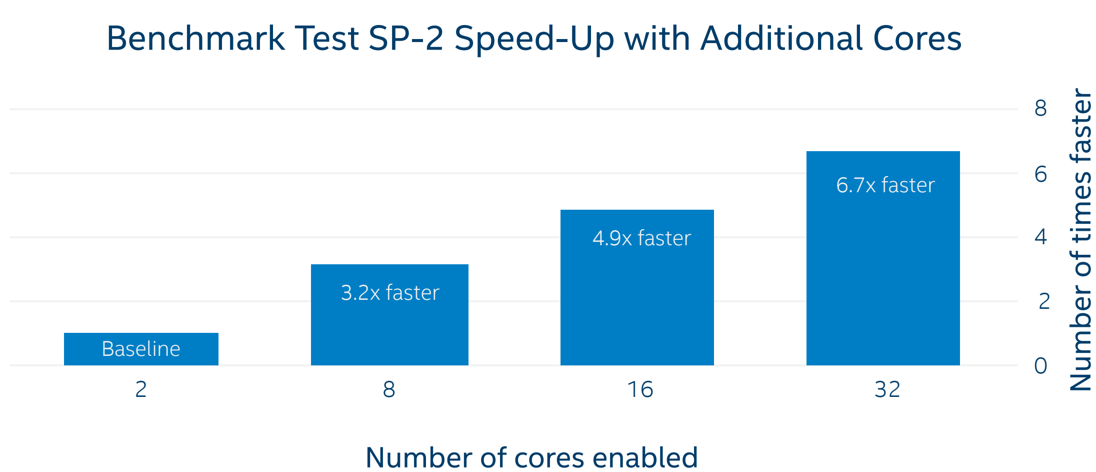

Figure 5. Performance improvements for benchmark test SP-2 as more cores were added[1]

Benchmark Test 3 of 5: Speaker (or "SP-3")

The ANSYS benchmark test SP-3 places a harmonic load on an acoustic speaker and its surroundings. Unlike the first two benchmark tests, this test performs *linear* FEA on the model because the properties of neither the speaker nor the surrounding air change as the harmonic load is placed on the model. Linear analysis is less CPU-intensive than nonlinear analysis, and it should therefore not be expected to show as much performance improvement with higher core counts as the previous two benchmark tests did.

Table 5 provides basic information about this benchmark test. Figure 6 shows both the original solid model (left) and the mesh (right) created in ANSYS Mechanical on which the test was performed.

Table 5. Basic information about benchmark SP-3

Analysis Type	Harmonic Linear Structural
Number of Degrees of Freedom	1,700,000
Equation Solver	Sparse
Matrix	Symmetric

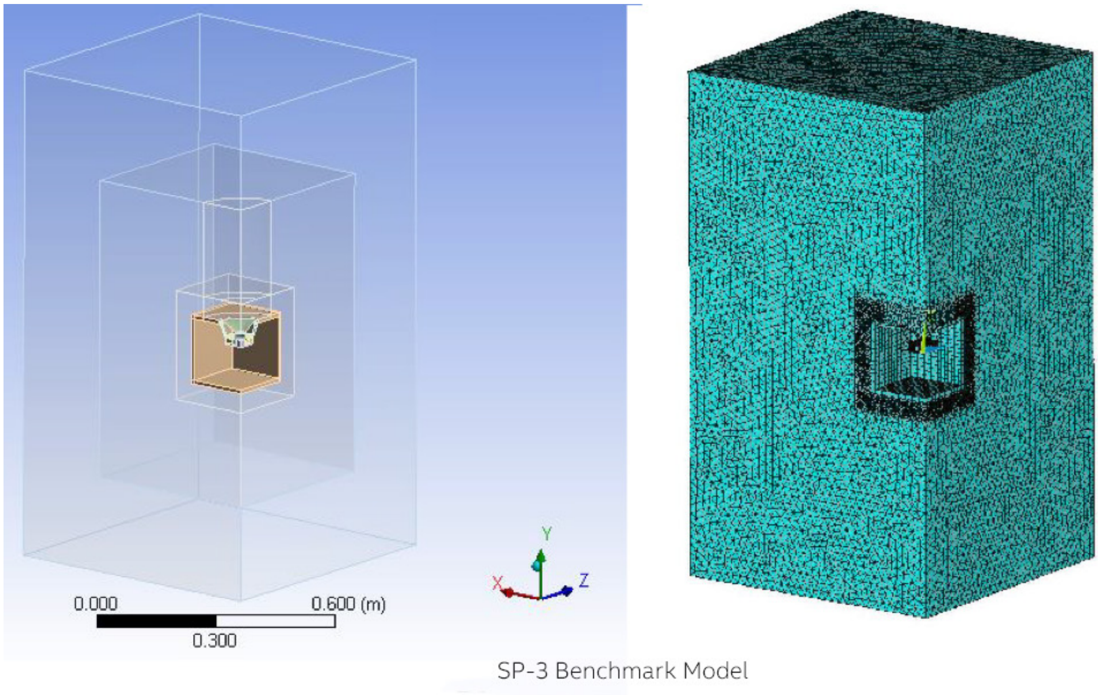

Figure 6. The SP-3 benchmark model before and after the meshing process

Benchmark Test 3 (SP-3) Results

With two cores enabled, the system completed the benchmark test SP-3 in 17 minutes and 28 seconds. Table 6 shows the full test results.[1]

Table 6. Results of benchmark test SP-3[1]

Cores enabled	Solution time
2	17 minutes, 28 seconds
8	6 minutes, 46 seconds
16	5 minutes, 24 seconds
32	4 minutes, 50 seconds

Figure 7 presents the performance-scalability improvements. Although performance with the highest core count (32) did improve by 3.6 times compared to the two-core system, this level of improvement is lower than those in the first two benchmark tests, as expected.

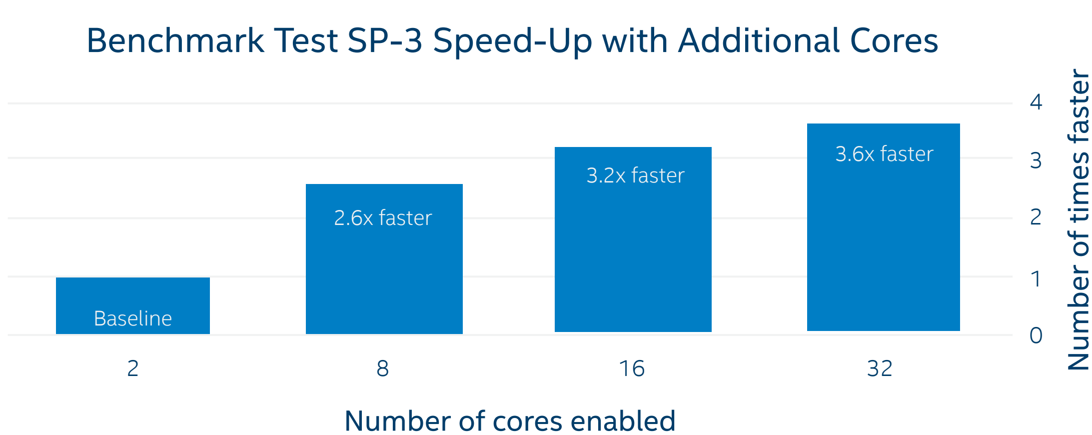

Figure 7. Performance improvements for benchmark test SP-3 as more cores were added[1]

Benchmark Test 4 of 5: Turbine (or "SP-4")

The ANSYS benchmark test SP-4 performs structural analysis of a turbine blade, such as one found in an aircraft engine. The analysis type used to solve this model is nonlinear and therefore CPU-intensive. Table 7 provides basic information about the test. Figure 8 shows the solid model (left) and the mesh (right) of the turbine blade.

Table 7. Basic information about benchmark SP-4[1]

Analysis Type	Static Nonlinear Structural
Number of Degrees of Freedom	3,200,000
Equation Solver	Sparse
Matrix	Symmetric

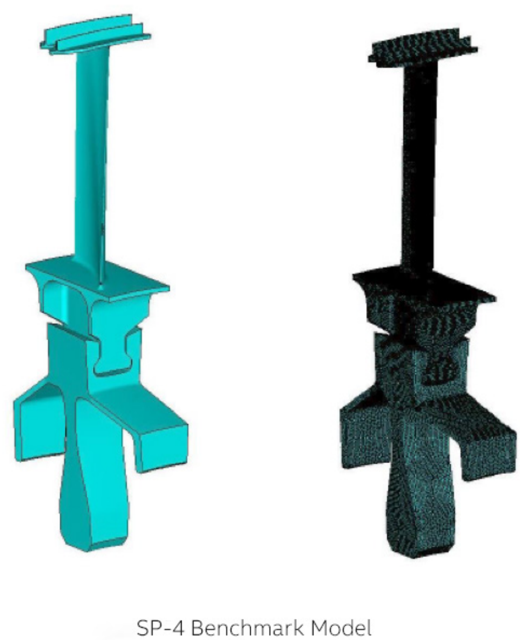

Figure 8. The solid and mesh used in benchmark test SP-4

Benchmark Test 4 (SP-4) Results

With two cores enabled, the Intel Xeon processor–based system completed benchmark test SP-4 in 19 minutes and 2 seconds. Table 8 shows the complete test results.[1]

Table 8. Results of benchmark test SP-4[1]

Cores enabled	Solution time
2	19 minutes, 2 seconds
8	6 minutes, 32 seconds
16	4 minutes, 26 seconds
32	3 minutes, 13 seconds

Figure 9 displays the performance scalability of the higher core counts. The solution time associated with the highest core count of 32 was almost six times faster than the solution time with only two cores enabled. This rate of performance improvement is close to that of the first two benchmark tests, which also performed nonlinear analysis.

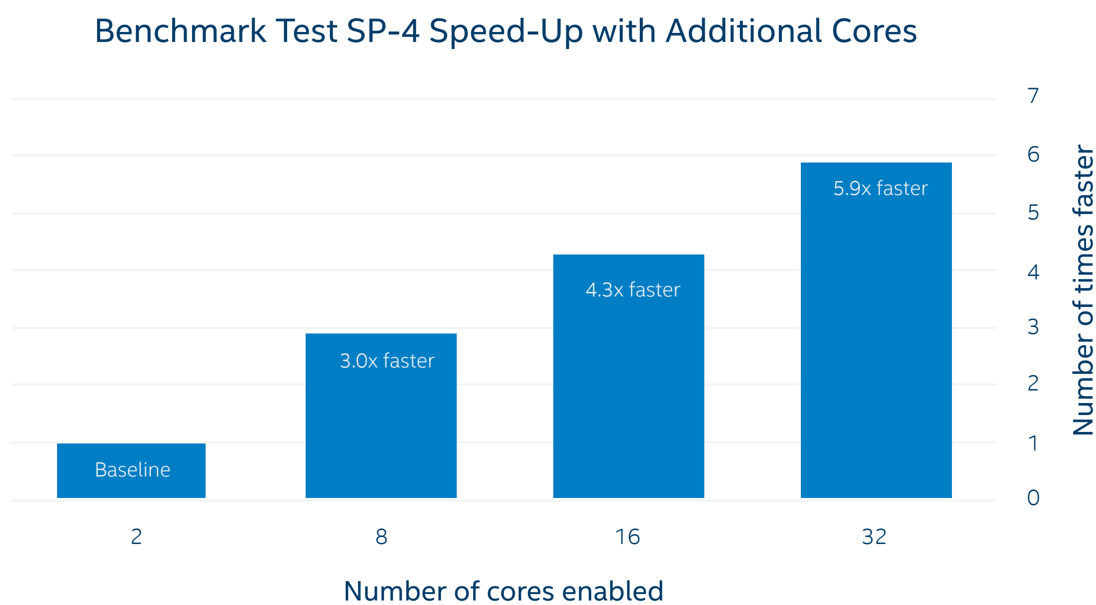

Figure 9. Performance improvements for benchmark test SP-4 as more cores were added[1]

Benchmark Test 5 of 5: Tractor Rear Axle (or "CG-2")

ANSYS benchmark test CG-2 performs structural analysis of a tractor's rear axle. Unlike the previous benchmark tests, the equation solver used for this test is iterative, which means that it completes the solution in multiple iterations. The analysis is nonlinear. Table 9 provides basic information about the test, and Figure 10 shows the solid and mesh used.

Table 9. Basic information about benchmark test CG-2

Analysis Type	Static Nonlinear Structural
Number of Degrees of Freedom	12,300,000
Equation Solver	Iterative
Matrix	Symmetric

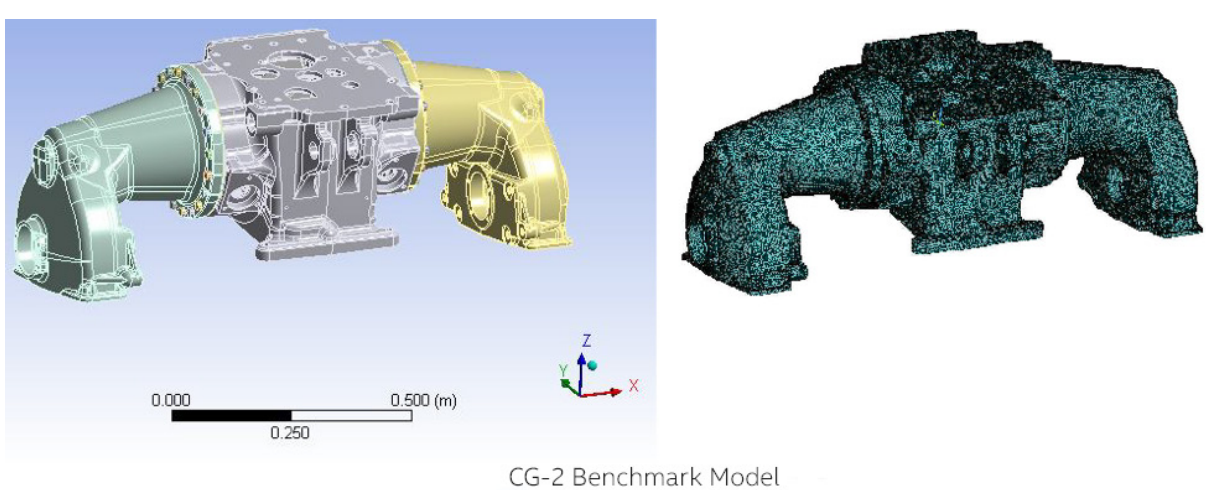

Figure 10. The solid and mesh used in benchmark test CG-2

Benchmark Test 5 (CG-2) Results

When two cores were enabled, the system completed the benchmark test in 43 minutes and 58 seconds. Table 10 shows the complete set of test results.[1]

Table 10. Results of benchmark test CG-2[1]

Cores enabled	Solution time
2	43 minutes, 58 seconds
8	11 minutes, 18 seconds
16	7 minutes, 13 seconds
32	5 minutes, 36 seconds

Figure 11 shows the plot of performance improvements with additional cores. The results of this test show the best rates of improvement for the higher-core-count systems out of any of the benchmark tests performed. These improvements nonetheless remain in the same general range as those of the other tests for which nonlinear analysis was performed. In this case, the system with 32 cores performed at a rate that was nearly eight times faster than the two-core system. At this level of performance scalability, a complex problem that would take 7 hours to solve on a two-core system would take a little over 53 minutes on the 32-core system.

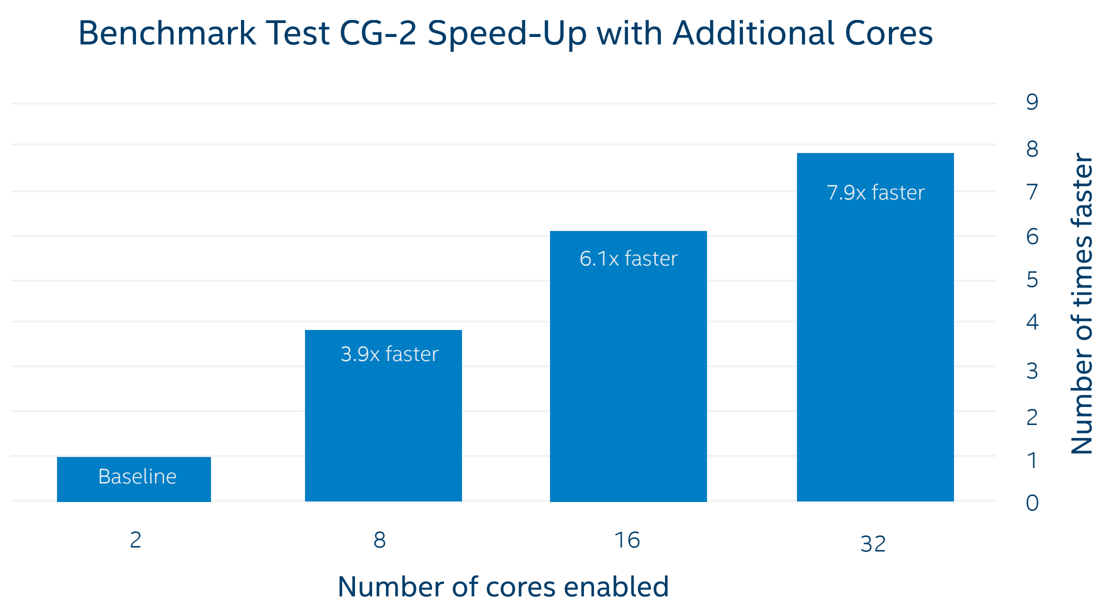

Figure 11. Performance improvements for benchmark test CG-2 as more cores were added[1]

Higher-Core Intel® Xeon® Processors Greatly Improve Engineer Productivity

ANSYS Mechanical has been developed to take advantage of the multiple cores in Intel® processors, and this advantage can be seen in the benchmark test results. The ANSYS Mechanical benchmark tests show significant improvements in solution times for FEA work performed with higher-core-count Intel Xeon processors. For the four benchmark tests requiring nonlinear analysis, the 32-core system performed at an average rate that was nearly seven (6.91) times faster than a system running on two CPU cores.

Because certain FEA operations can be extremely time-consuming and CPU-intensive, the choice of core count for an engineering workstation's CPU can result in dramatic, real-life changes for engineers. After migrating to a higher-core-count system, an engineer can complete what used to be an overnight simulation during a lunch break instead, and an engineer can complete what used to be a lunch-break simulation during a coffee break instead. For engineers who save five hours just once every three weeks, that time saved accumulates to more than 80 saved hours per year, or more than two entire work weeks.[1]

Beyond the time saved through faster analysis, another advantage of a higher-core-count system is that it allows engineers to complete all stages of the FEA workflow on a single system without needing to export mesh files to a higher-end workstation or cluster in order to perform the solution phase. This, by itself, prevents interruption of the workflow and allows engineers to complete their work faster.

Yet another benefit of a faster system is the potential for improved product quality. With faster testing, engineers can perform more and different stress tests in the same amount of time. This provides an opportunity to conduct more thorough stress testing, which can result in safer and more durable structures.

To support engineer productivity, minimize wait times, meet more deadlines, improve structural quality, and speed the time-to-market for engineered products, anyone considering the purchase of a CAE system should strongly favor systems with higher-core-count CPUs. The performance gains and other business benefits offered by higher-core-count systems can result in savings over time that can more than compensate for any additional upfront costs and ANSYS licensing fees. And active specials on ANSYS licensing can make higher core counts especially attractive in terms of the cost-benefit ratio.[2]

In the future, Intel will continue to increase core densities in new Intel Xeon processors and will offer additional innovations to help speed the performance of ANSYS software. ANSYS, for its part, will continue to work together with Intel to optimize its software for the increasing core counts and other acceleration technologies that are offered by Intel.

Learn More

To learn more about professional workstations powered by Intel® processors, visit **intel.com/content/www/us/en/workstations/workstation-products.html**.

To find out more about ANSYS® products, visit **ansys.com/products/structures/hpc-structures**.

To learn more about how Intel technologies speed the performance of ANSYS products, visit **ansys.com/intel**.

[1] Testing by ANSYS in 2017. Configuration: CPU: 2-socket, 22-core Intel® Xeon® processors E5-2699 v4 (2.2/3.6 GHz, 110 MB Smart Cache). Graphics: NVIDIA Quadro M4000*. Memory: 64 GB DDR4 RAM. Storage: two 400 GB Intel® Solid-State Drive (SSD) 750 Series Peripheral Component Interconnect Express* (PCIe*) drives with NVM Express* (NVMe*) configured in RAID 0. Operating system: Windows 10*.

[2] Contact an ANSYS representative for any questions you might have about ANSYS licensing: https://pages.email.ansys.com/contactme/?&e=Contact&t=ContactMe&d=Contact Me&tsd=ContactTrigger-EN&p=ANSYS Licensing

Printed in USA 0817/KE/PRW/PDF Please Recycle 336174-001US