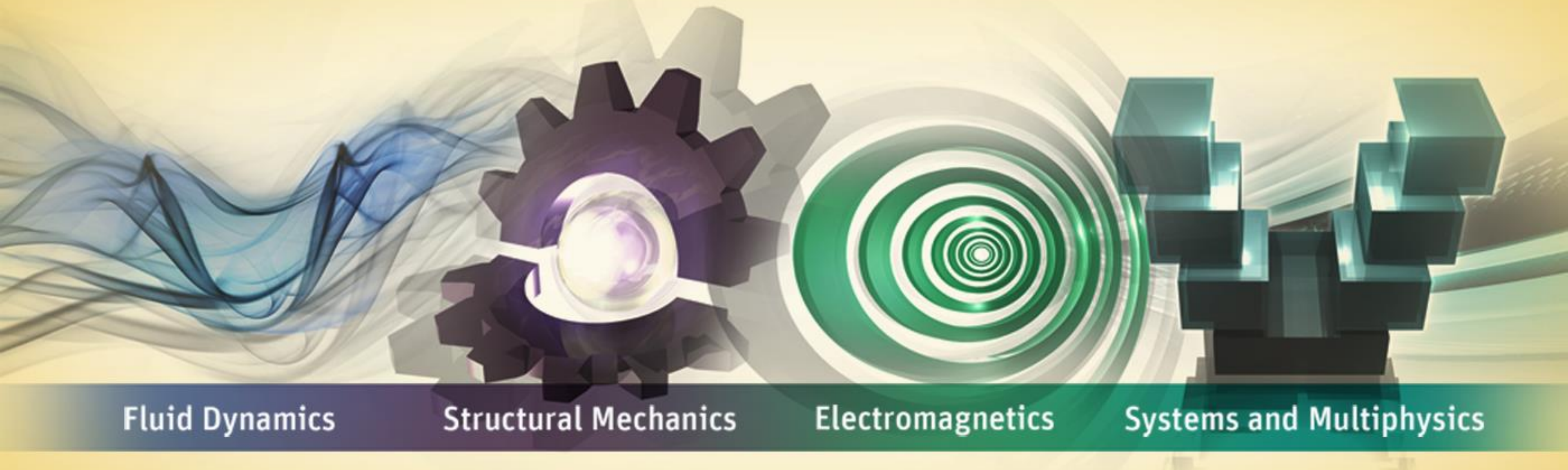
Realize Your Product Promise™

ANSYS HPC computing

effective source allocation

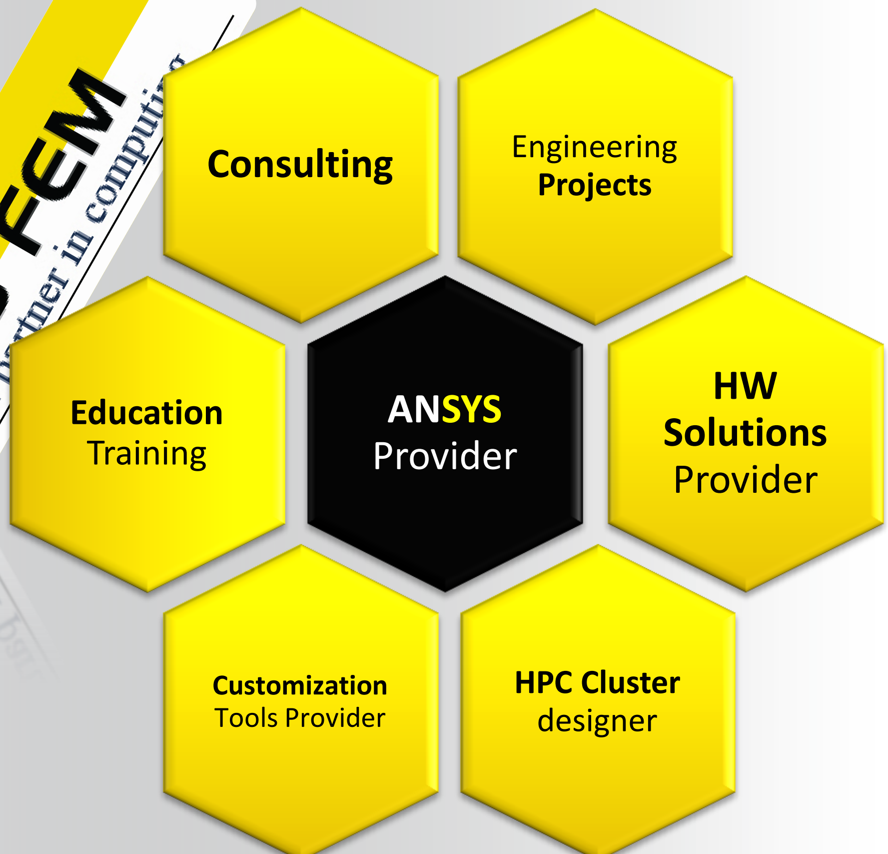


Petr Koňas

SVS FEM
Your partner in computing

Content

- Who is SVS FEM?
- What is ANSYS?
- ANSYS in Academic World
- Comparison ANSYS CFX and OpenFOAM
- HPC Configuration – How to select HW resources for your job?
- ANSYS Cloud Tools
- Solution of Large models (Superelements)
- Benchmarks of large clusters (Metacentrum, IT4I, UV2000)



SVS FEM is the original **ANSYS Channel partner** for Czech Rep. and Slovak Rep. since 1992 and partner of **CADFEM GmbH.**

ISO 9001:2009

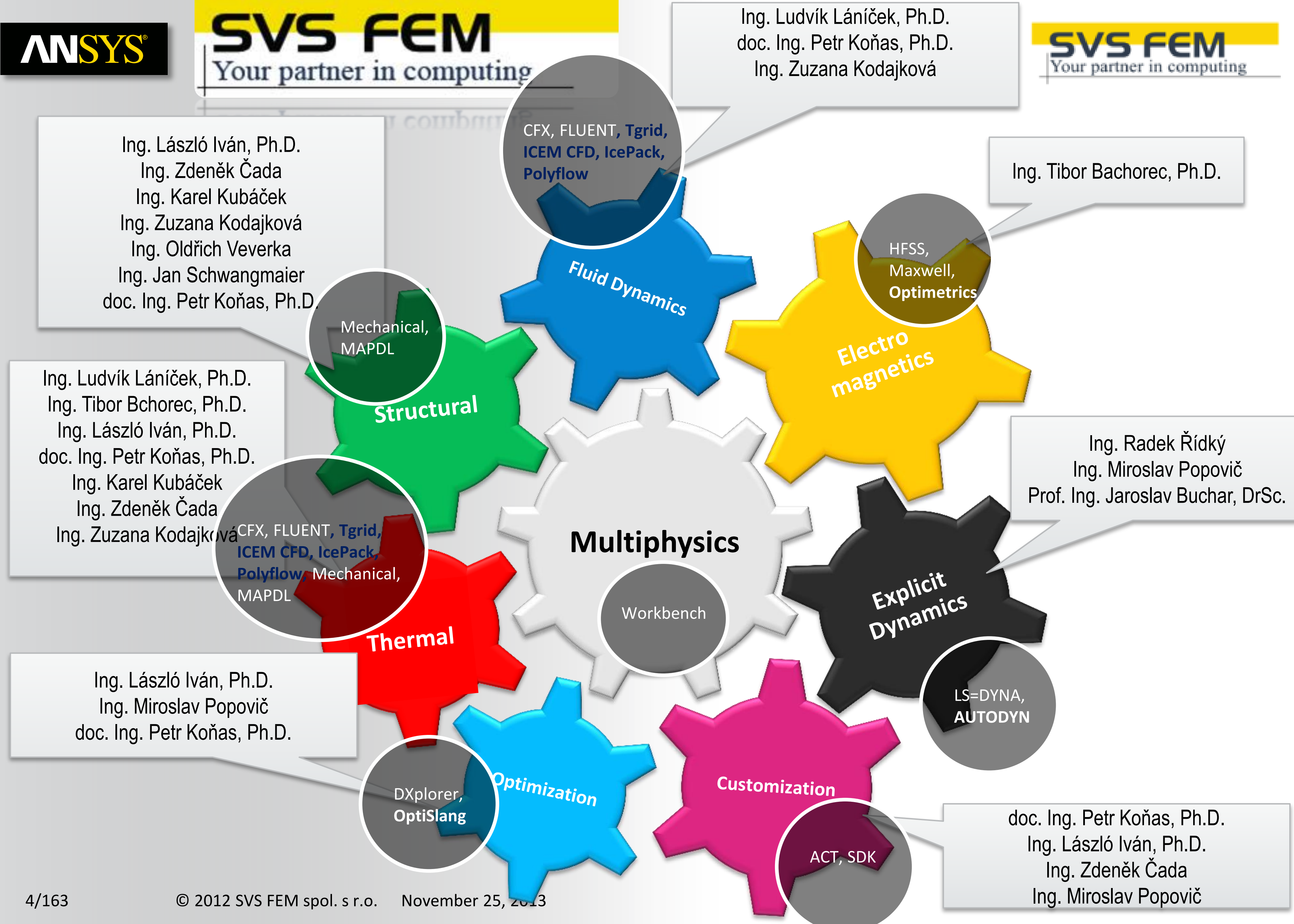
A Member of TechNetAlliance

NAFEMS

EDGE ALLIANCE

SVS FEM

Your partner in computing



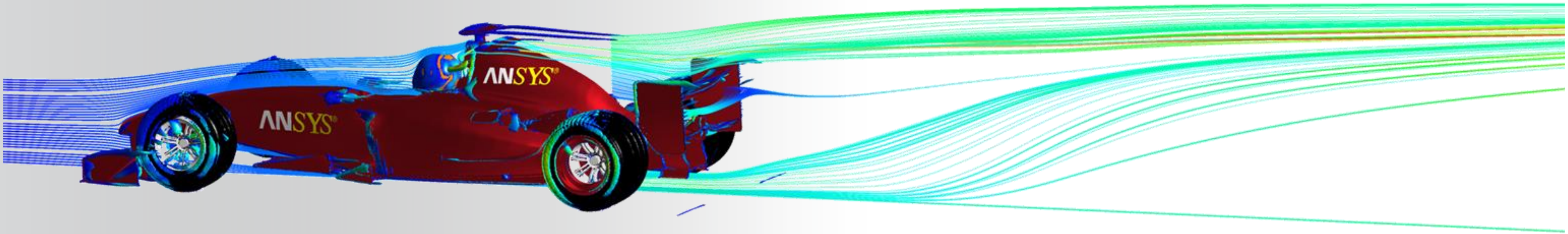
Školení SVS FEM

SVS FEM
Your partner in computing

- **ANSYS Mechanical APDL 1**
- **ANSYS Mechanical APDL 2**
- ANSYS Mechanical APDL - **Dynamics**
- ANSYS Mechanical APDL - **Nonlinear 1**
- ANSYS Mechanical APDL - **Nonlinear 2**
- ANSYS Mechanical APDL - **Nonlinear 3**
- ANSYS Mechanical APDL - **Nonlinear 4**
- ANSYS Mechanical APDL - **Optimalization**
- ANSYS Mechanical APDL - **Programming**
- ANSYS Mechanical APDL - **Emag NF**
- ANSYS Mechanical APDL **- Emag HF**
- **ANSYS CFX**
- **ANSYS FLUENT**
- **ANSYS Icepak**
- **ANSYS from FLUENT to CFX**
- **ANSYS TurboGrid**
- **ANSYS BladeModeler**
- **ANSYS CFD-Post**
- **ANSYS ICEM CFD**
- **Gambit to DesignModeler & ANSYS Meshing transition**
- **ANSYS CFX - Spalování a Radiace**
- **ANSYS FLUENT - Spalování a Radiace**
- **ANSYS CFX FSI**
- **ANSYS FLUENT FSI**
- **ANSYS Workbench Mechanical 1 / ANSYS DesignSpace**
- **ANSYS Workbench Mechanical 2**
- ANSYS Workbench Mechanical - **Dynamics**
- ANSYS Workbench Mechanical - **Nonlinear 1**
- ANSYS Workbench Mechanical **- Nonlinear 2**
- ANSYS Workbench Mechanical **- Nonlinear 3**
- ANSYS Workbench Mechanical - **Thermal**
- ANSYS Workbench Mechanical - **Programming**
- ANSYS Workbench Mechancial - **Fatigue Modul**
- ANSYS Workbench Mechanical **- nCode DesignLife**
- ANSYS Workbench Mechanical - **Emag NF**
- ANSYS Workbench **Geometry (ANSYS DesignModeler)**
- ANSYS Workbench **Geometry (ANSYS SpaceClaim Direct Modeler)**
- ANSYS Workbench **DesignXplorer**
- ANSYS Workbench **Explicit Dynamics**
- **Elektromagnetické simulace - Maxwell**
- **Elektromechanické simulace - Maxwell, Simplorer, RMxprt**
- **ANSYS AUTODYN**
- **LS-DYNA**
- Školení OptiSlang

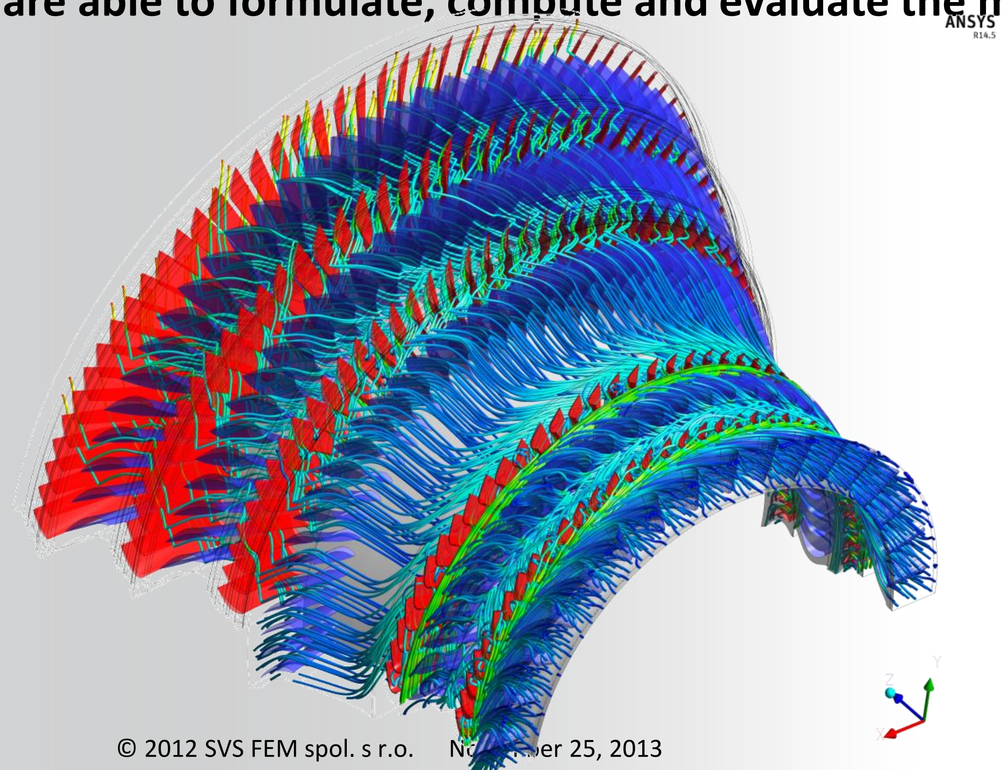
http://www.svsfem.cz/content/plán-školení-pro-rok-2013

We can solve Extreme Models



140M cells

We are able to formulate, compute and evaluate the most complex and the largest models…



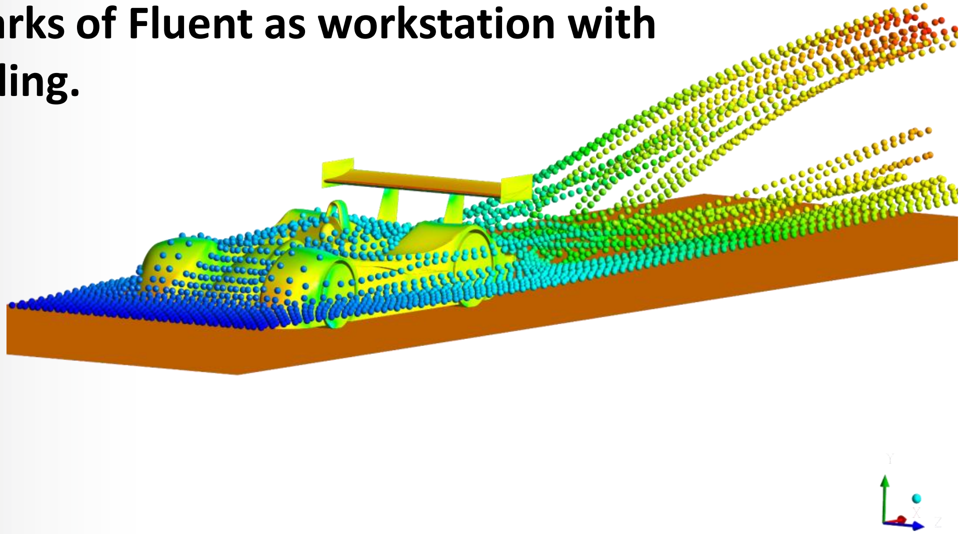
280M cells

ANSYS

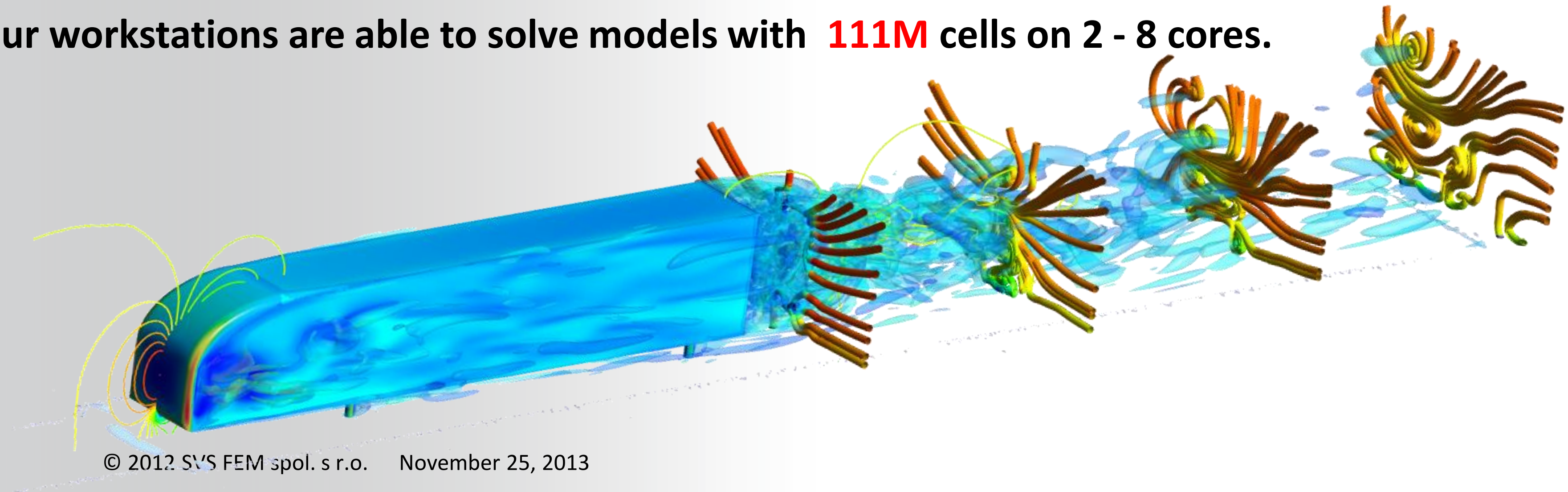
We can build infrastructure for HPC

SVS FEM
Your partner in computing
ANSYS

- **Our Workstation won in official benchmarks of Fluent as workstation with the best power per core and the best scaling.**



- **Our workstations are able to solve models with 111M cells on 2 - 8 cores.**



Pro podporu našich uživatelů i po stránce HW stala se SVS FEM s.r.o. oficiálním partnerem firem ABACUS, HP, SGI a IBM.

Pro každý ze SW produktů u nás zakoupených dokážeme navrhnout optimální hardware a nabízíme dodání celé instalace na klíč.

Hardware od SVS FEM

SVS FEM je váš specialista na CAE Hardware. Od přípravy přes konfiguraci, instalaci až po vlastní užívání.

SVS FEM je Váš partner ve výpočtech

sgi CHANNEL PARTNER

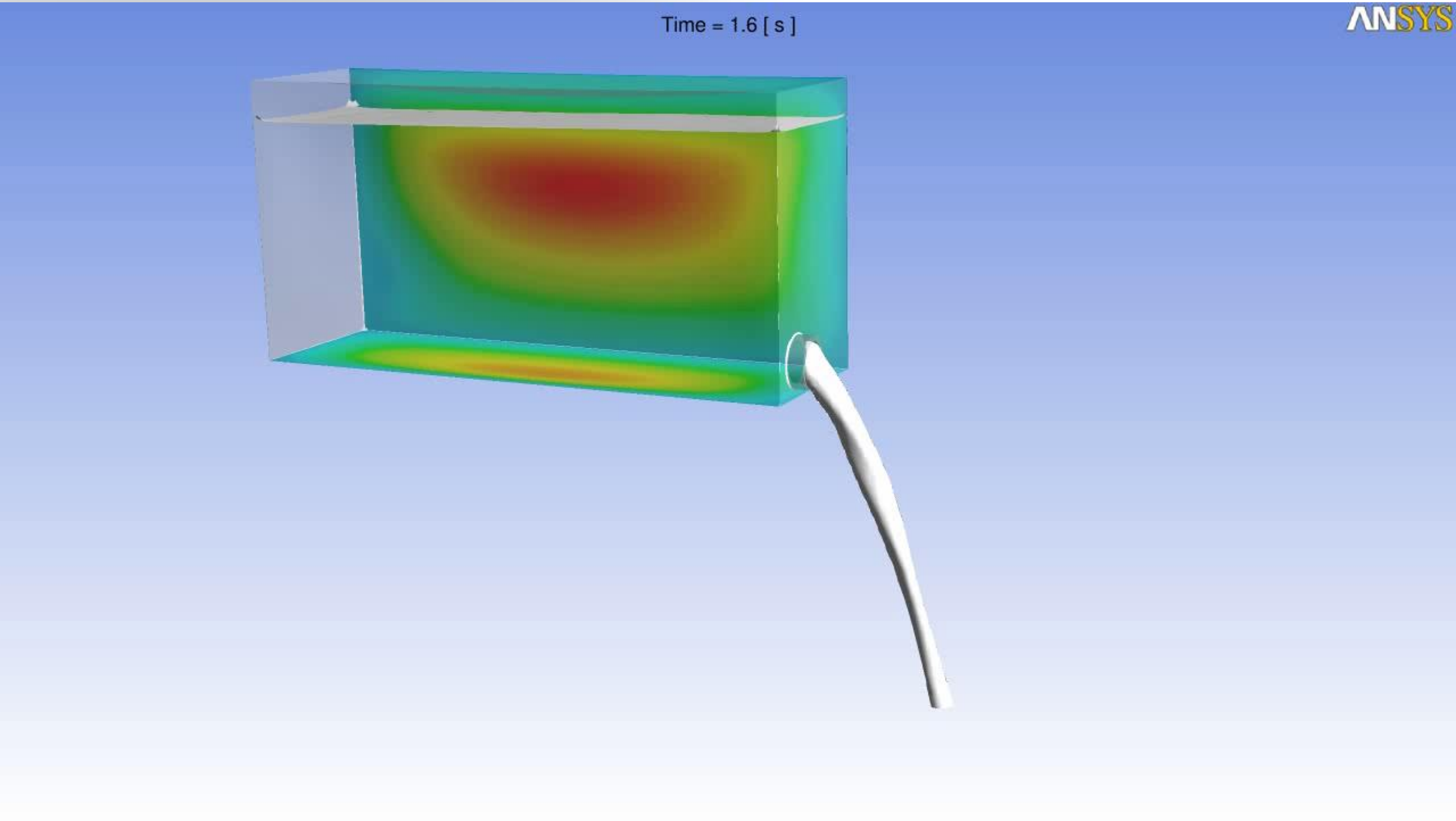
- **Pracovní stanice**
- **Notebooky**
- **Servery**
- **Clustery**
- **Komponenty a příslušenství**

Proč zvolit právě naši konfiguraci HW a instalaci ANSYSu?

- Sestavujeme pouze takový hardware, který je kompatibilní s produkty ANSYS a který dosahuje optimálního výkonu dle Vašich potřeb
- Provádíme kontrolu kvality dodávaných PC vůči stabilitě ANSYSu
- Instalujeme požadovaný operační systém i balíky ANSYSu včetně nejnovějších aktualizací. Provedeme tunning profilu testovacího uživatele.
- Testujeme/aktualizujeme ovladače BIOSu a provádíme jejich tunning pro maximální výkon ANSYSu
- Konfigurujeme Hardware dle vašich požadavků: RAID disků, firmware disků, obslužné rutiny, konfigurace/tunning síťových adapterů, automatický backup
- Konfigurujeme plně MPI (Platform, Open, Intel) pro maximální výkon při distribuovaném výpočtu u jednotlivých aplikací ANSYS
- Naši konfiguraci dodáváme včetně podrobného auditu.
    - Výpis profilu uživatele
    - Test kompatibility ANSYS kritérií
    - MPI test
    - Vytvoření geometrie, vysíťování a provedení řešení na jednoduché úloze
    - Středně těžké až extrémní CFX/Fluent/Mechanical benchmarky
    - Intenzivní HW benchmarky
    - Konfigurace RSM, test úloh přes RSM
    - Verifikace RAID polí, simulace selhání a následná obnova
    - Výsledky všech testů Vám zůstanou k dispozici na předávaném HW.
- Support (hotlin/webex/zásah na místě) v případě HW či SW problémů. Webex pro první krůčky v naší instalaci.

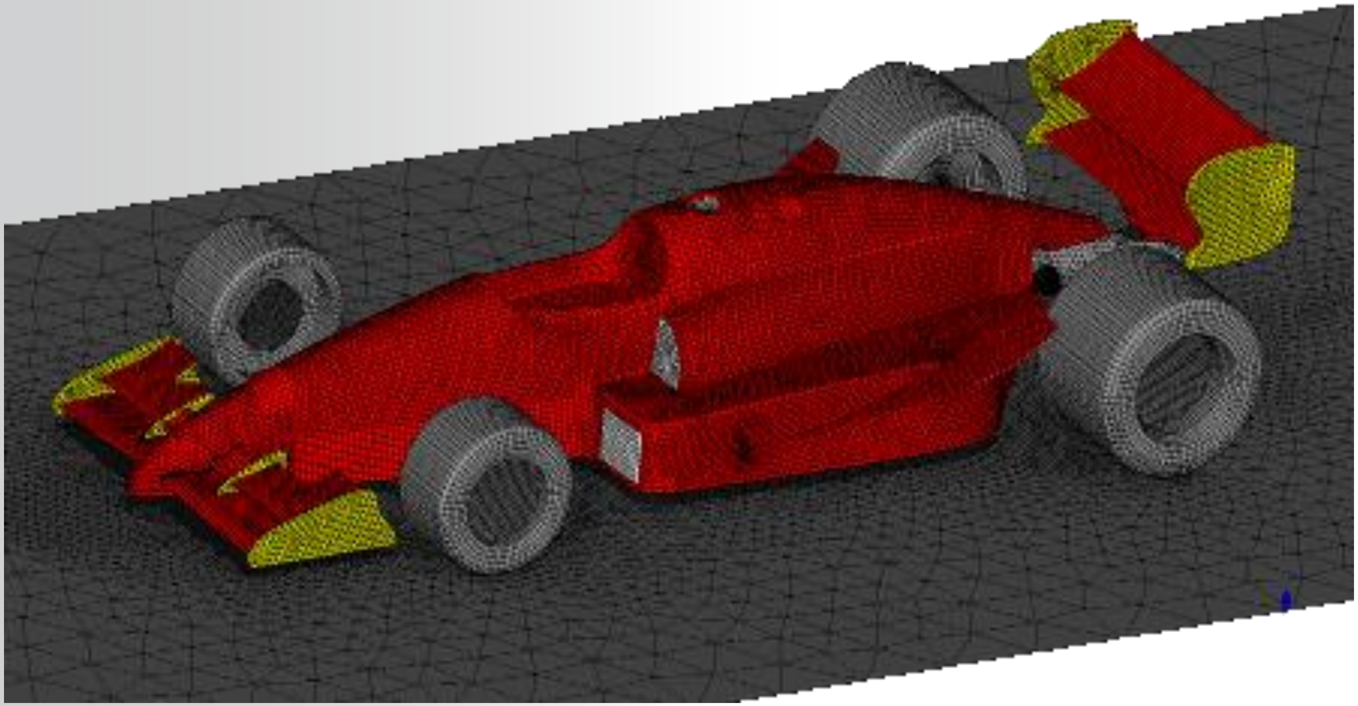
What is ANSYS?

Sofisticated engineering tool for solution of complex physical tasks and...



What is ANSYS?

...complex geometry of the real problems...



What Does ANSYS do?

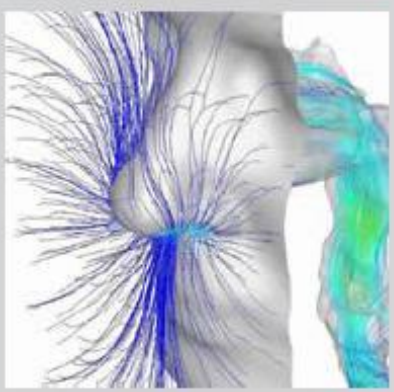
SVS FEM
Your partner in computing

Digitally simulate performance across all physics of complete systems, in their real-world environments

Concept Development — Evaluate New Ideas

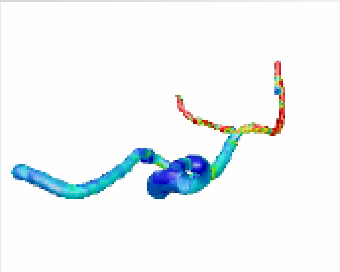
Feasibility Assessment — Assess Designs

Final Design — Engineer Manufacture



ANSYS Enables Simulations

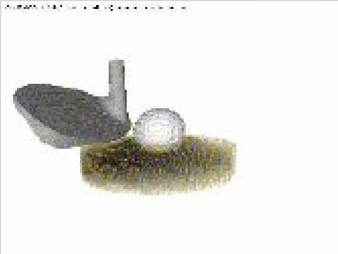
Automobile Crankshaft



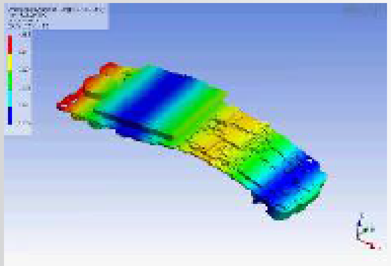
Biomed Fly Through

Robotic Arm

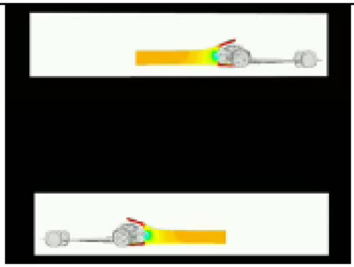
Production Equipment



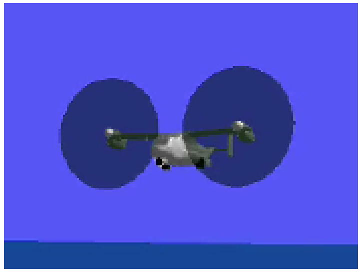
Golf Club/Ball Contact



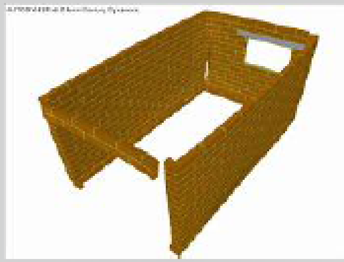
Cell Phone Stress



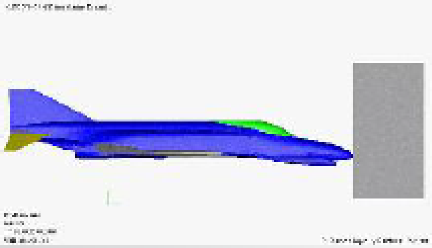
Engine Cooling



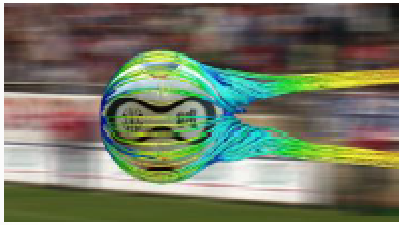
Aerospace



Building Explosion



Jet Impact

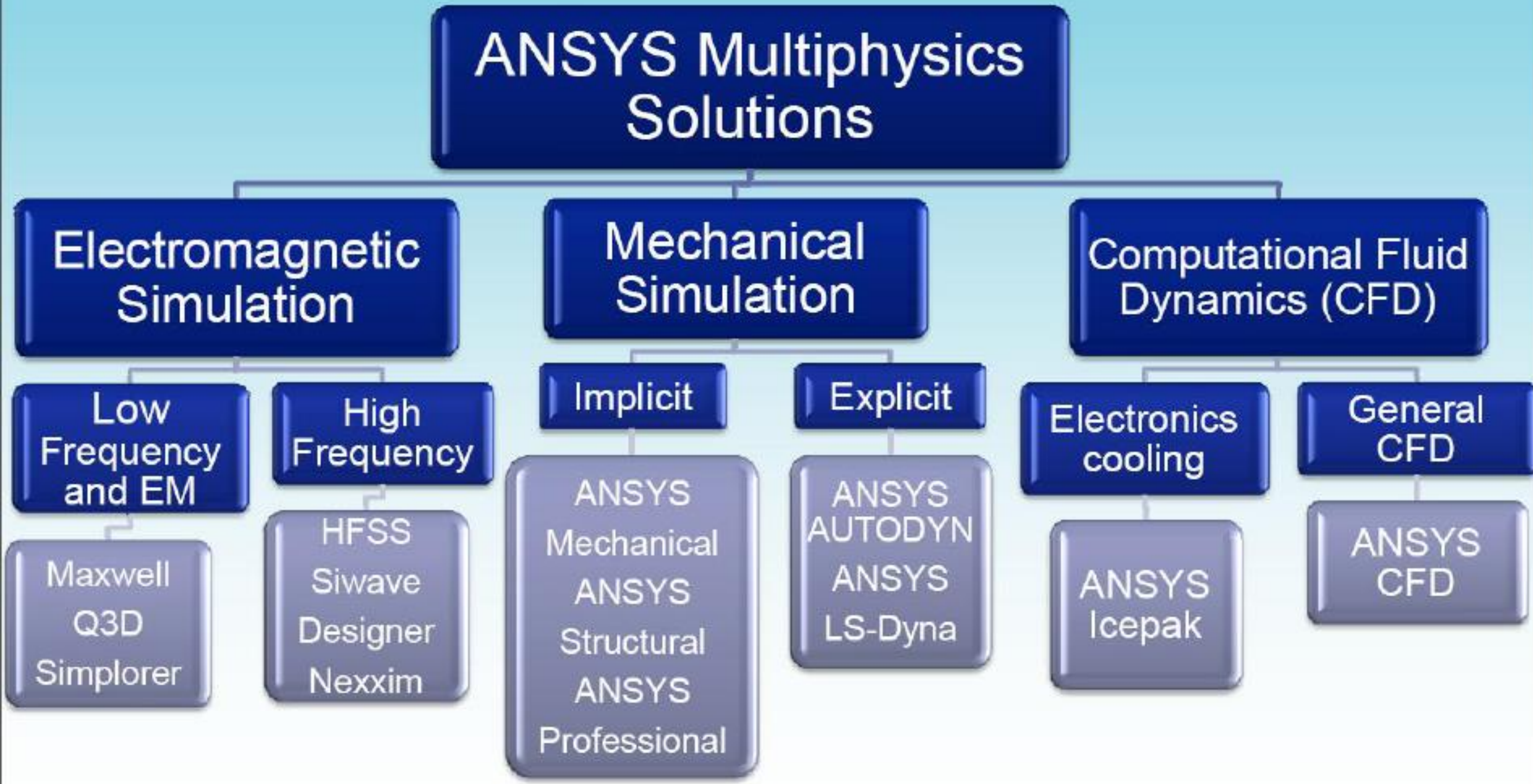


High Speed Airflow of Soccer Ball

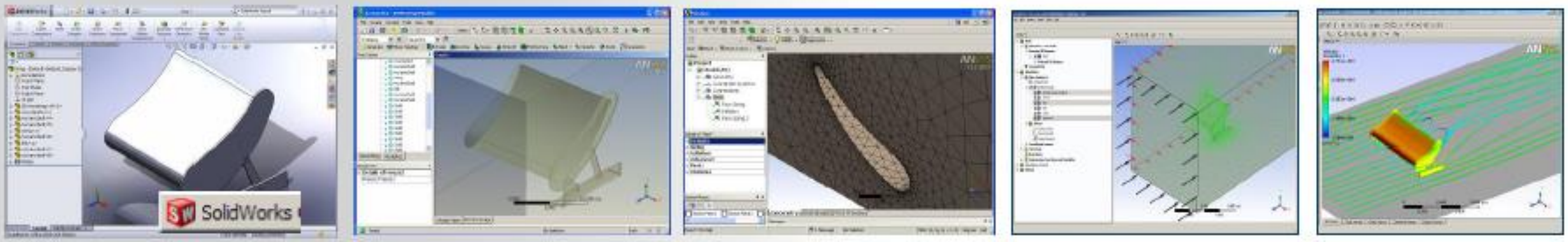
Cyclone Separator

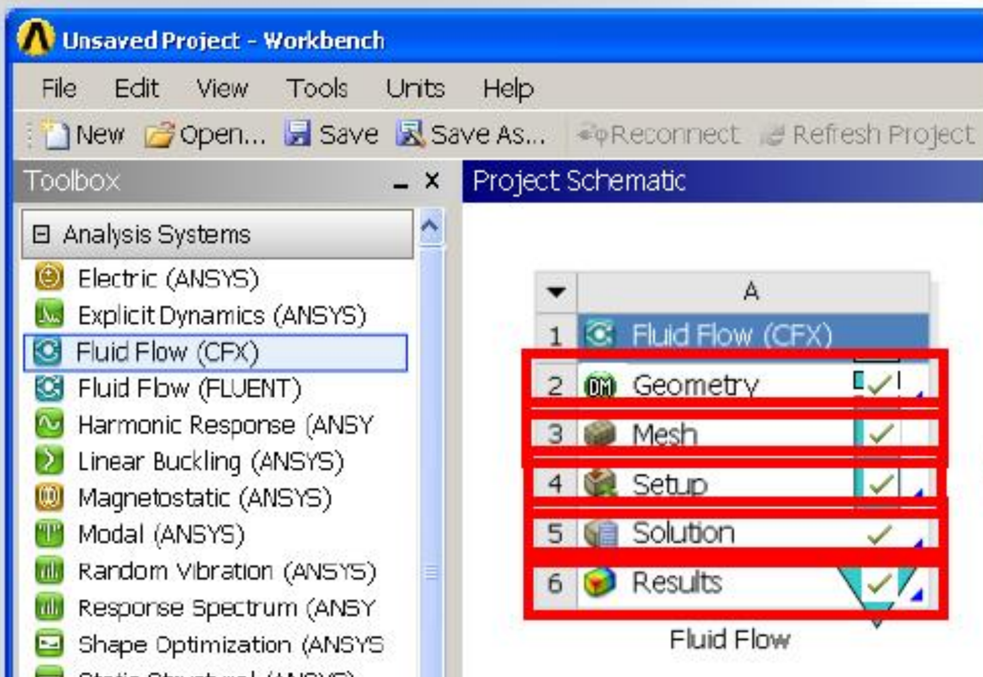
tesy of Gebr. Heller Maschinenfabrik GmbH and University of Sheffield

ANSYS Solutions



The Project Schematic



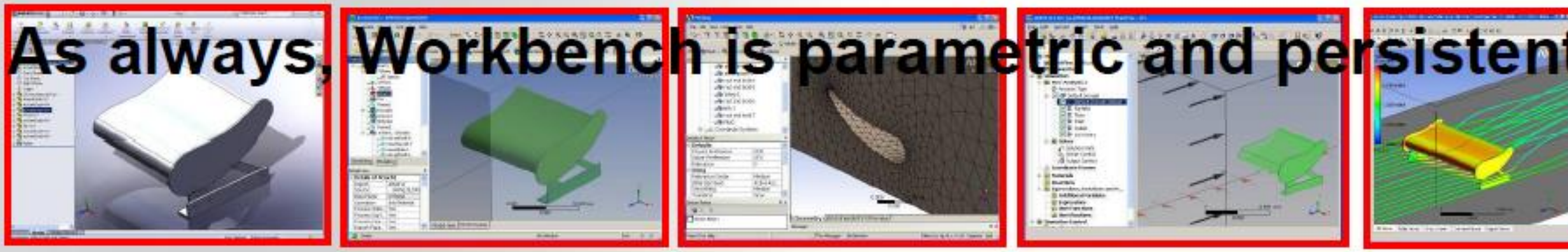


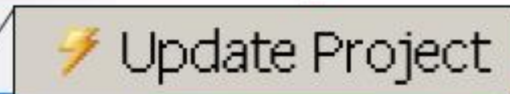
- Attach to CAD model
- Edit Geometry
- Generate Mesh
- Define Physics
- Solve
- Post-process

Automatic System Updates

SVS FEM
Your partner in computing

As always, Workbench is parametric and persistent





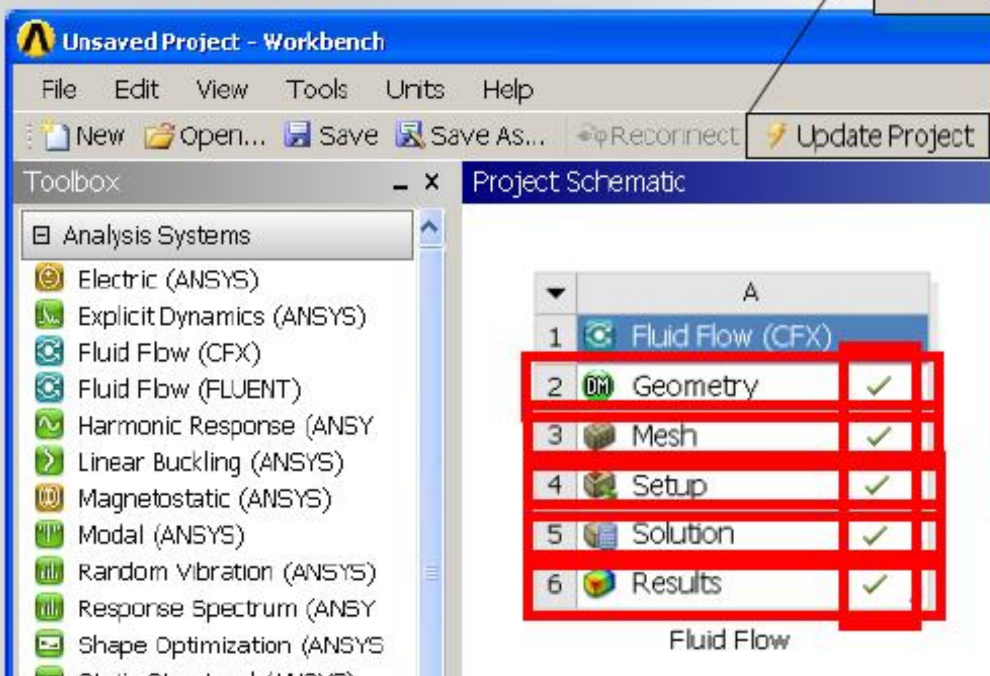
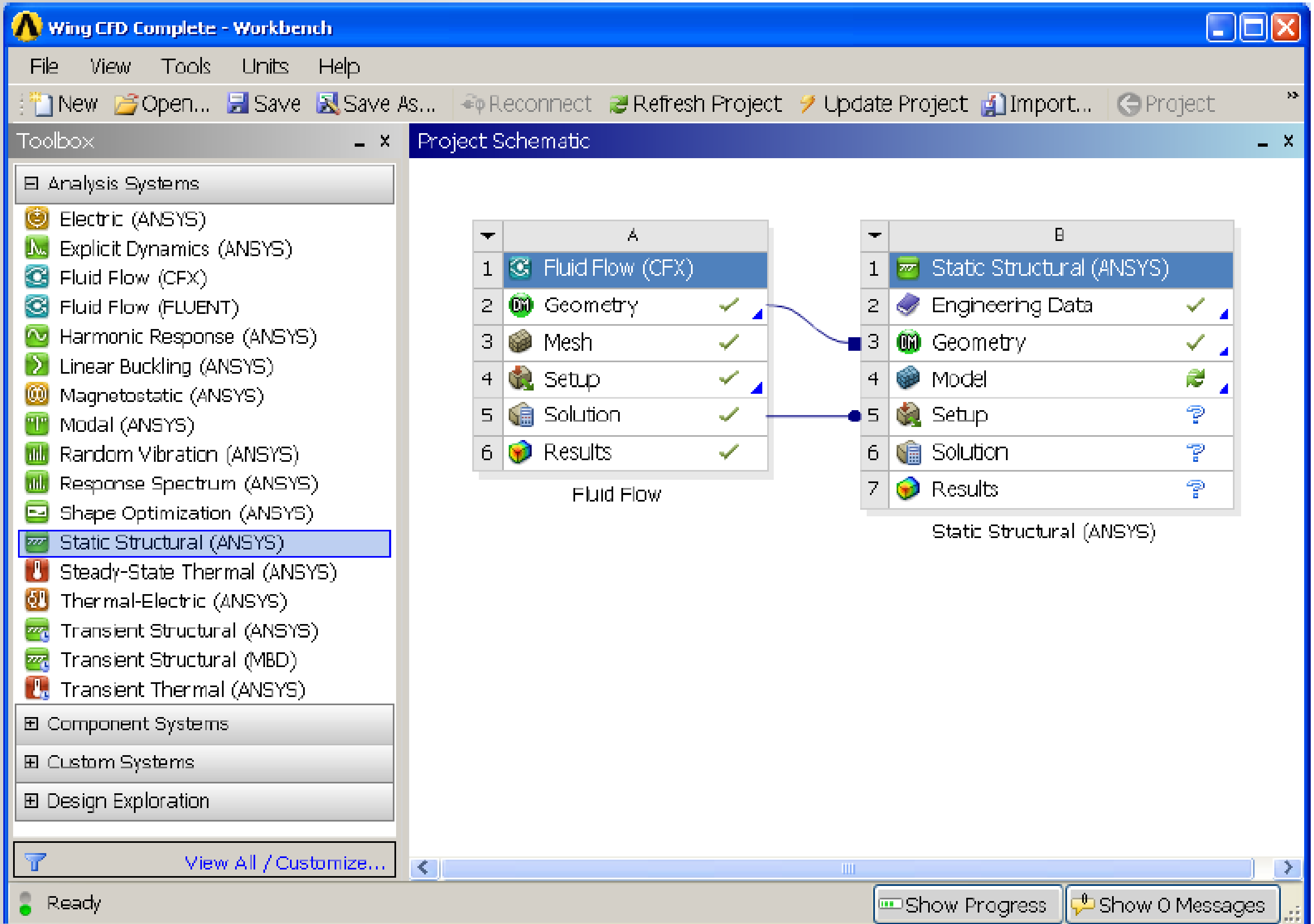


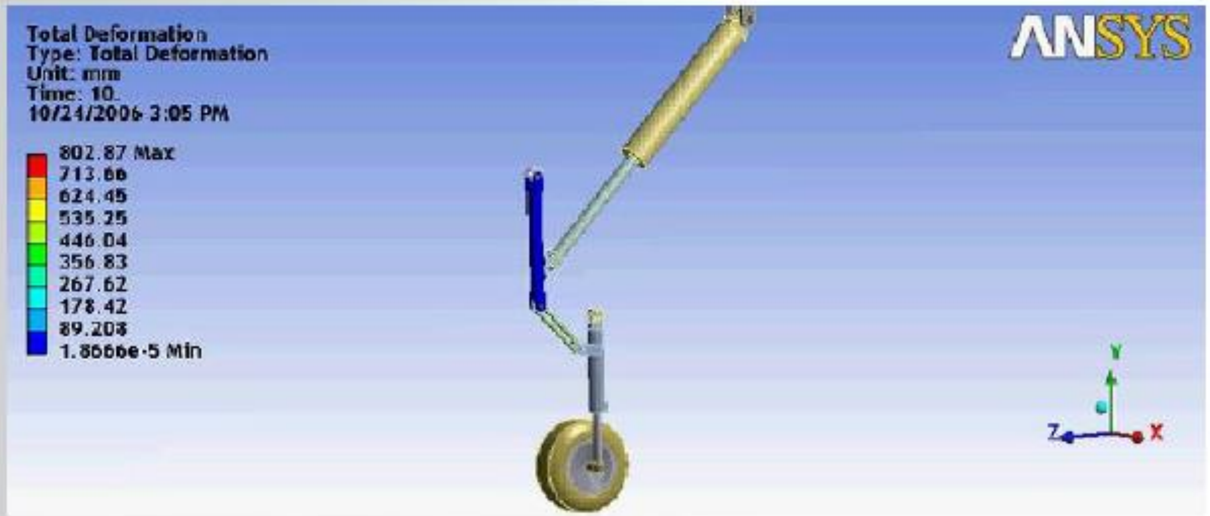
Illustration:

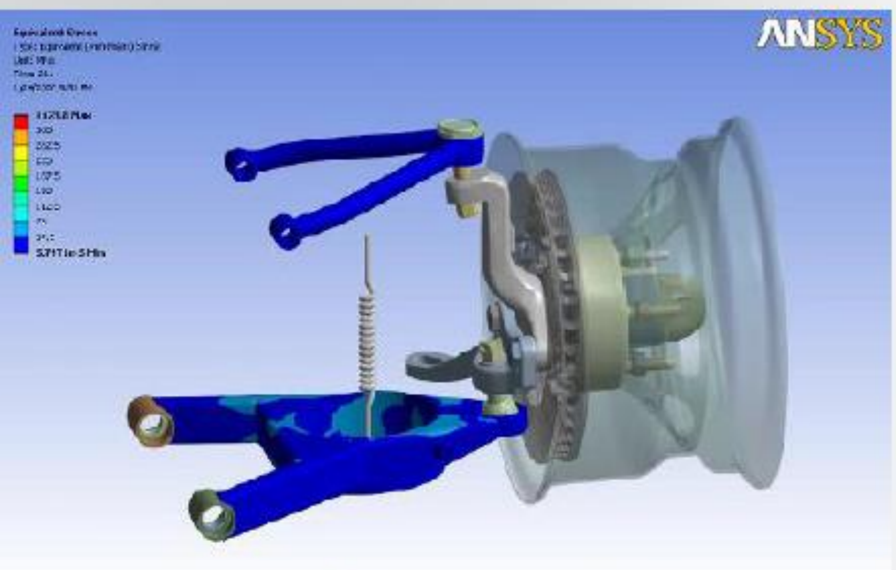
- Change the geometry
  - State icons change
- Update the project
- Entire project updates in batch mode

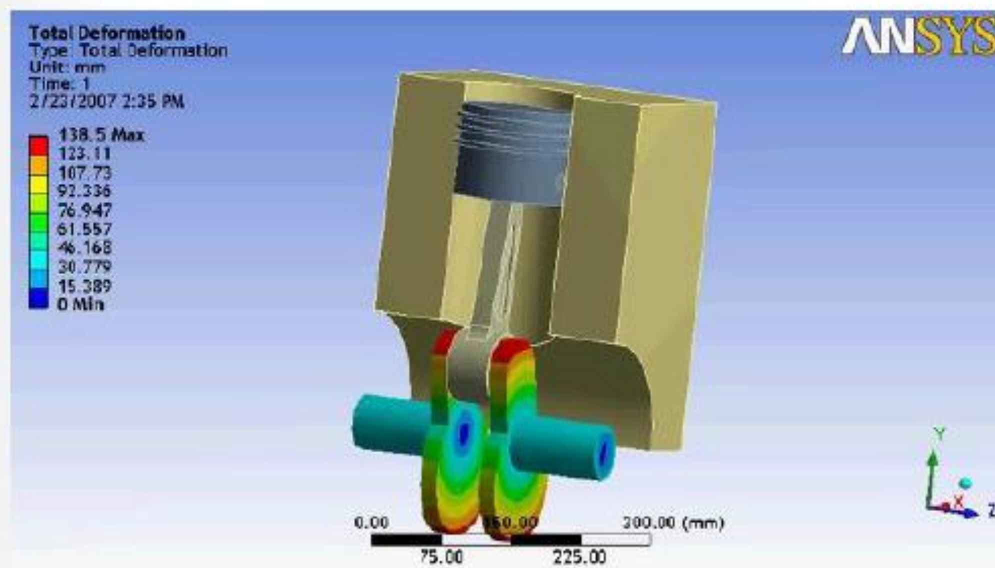
ANSYS Workbench Enables… Multiphysics Simulations



Flexible Multibody Dynamics

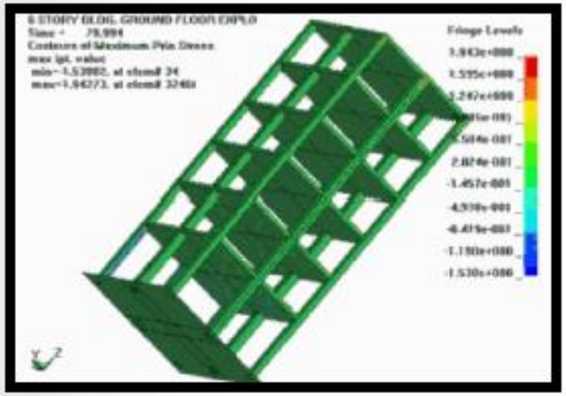


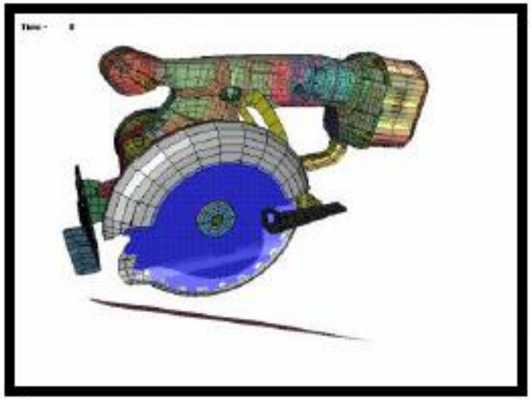


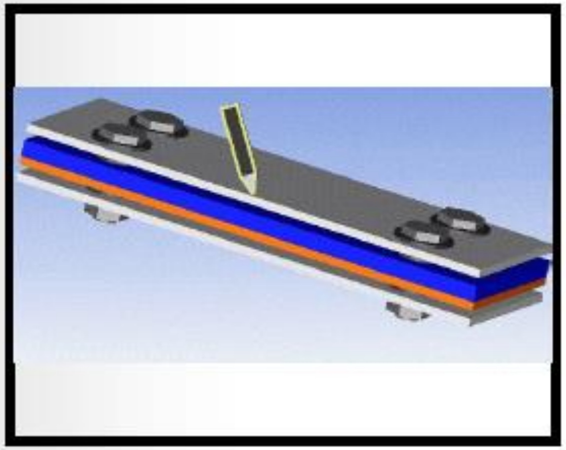


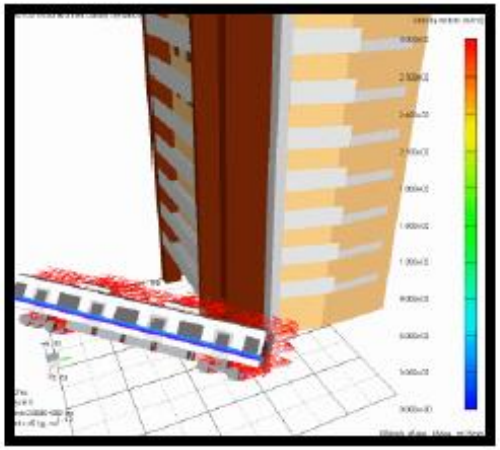
Explicit Dynamics Solutions

- Three complementary solutions: AUTODYN, LS-DYNA, and Explicit STR
- Workbench-based
- Interfaces to CAD systems
- Access to meshing tools
- Parametrically Associative to Geometry
  - Unique for Explicit!
  - Explicit Dynamics can be used as an up-front design tool for the first time

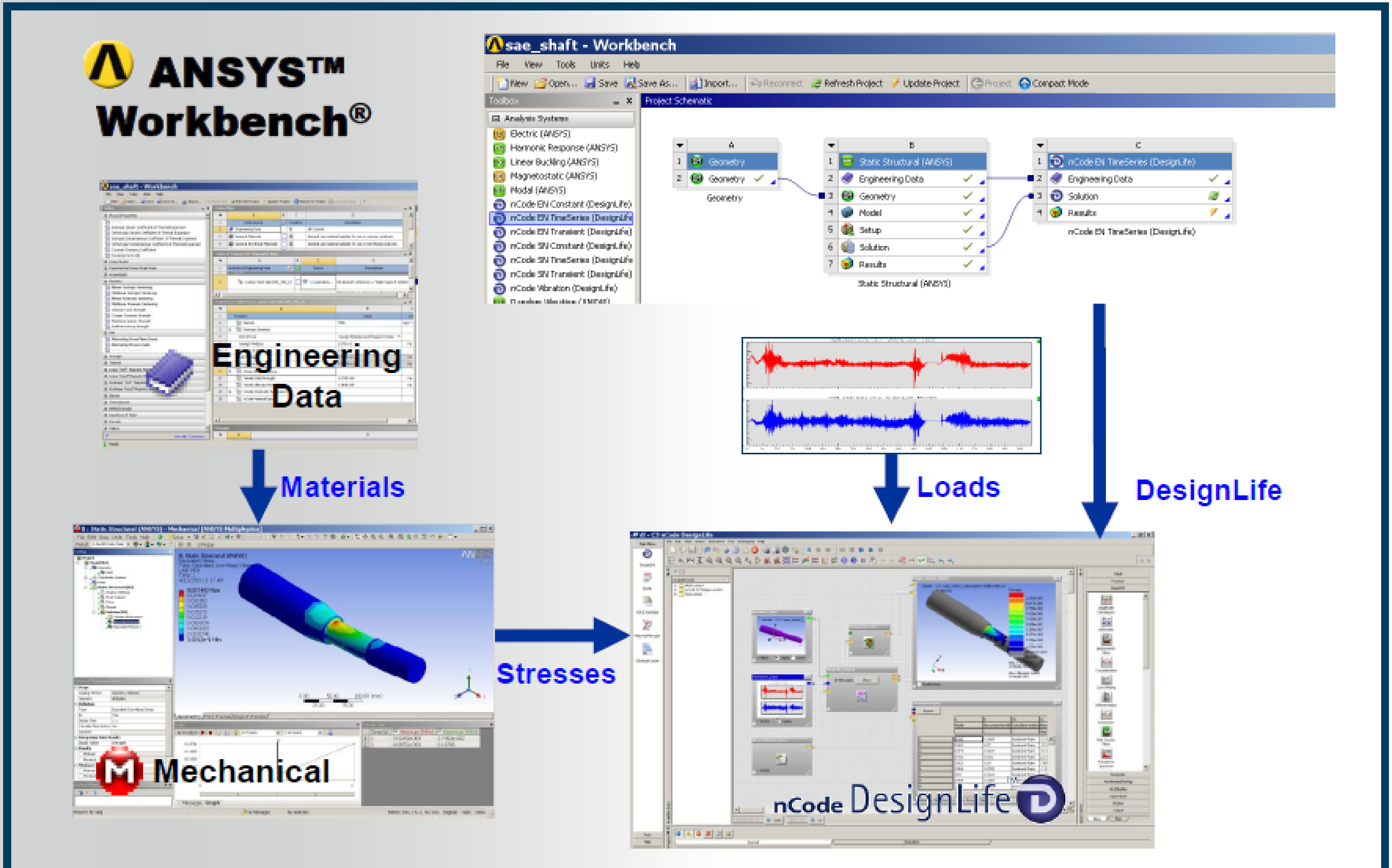








ANSYS® nCode DesignLife for Fatigue Analysis

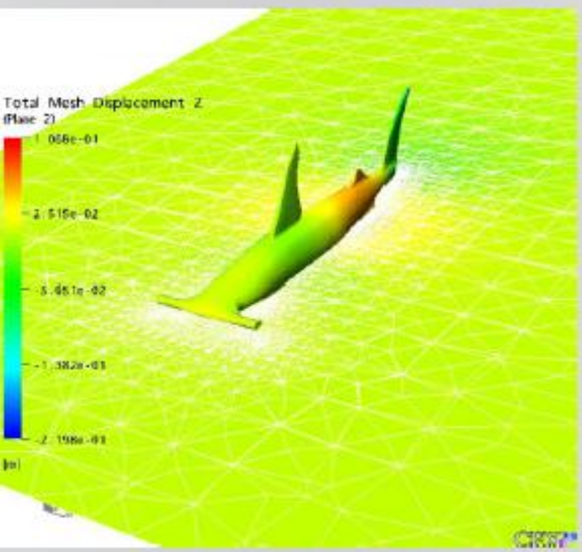


 November 25, 2013

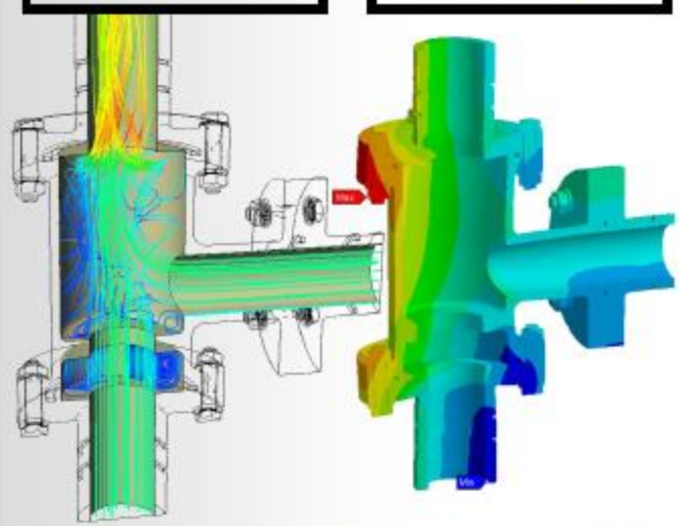
Fluid-Structure Interaction

SVS FEM
Your partner in computing

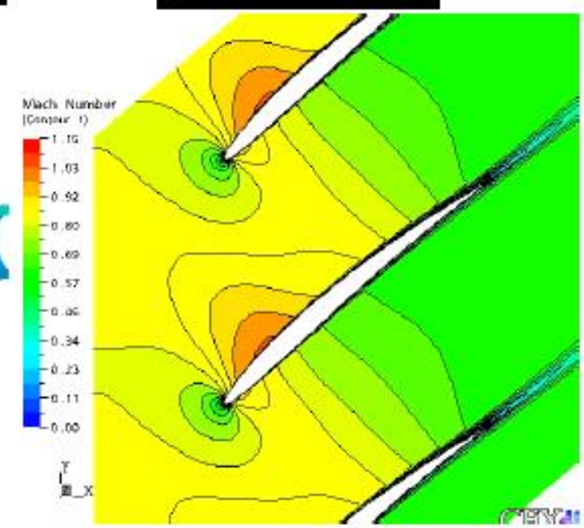
Direct FSI	Basic FSI		Advanced FSI
↓	ANSYS	ANSYS	ANSYS
	↑	↓	↕
CFX / FLUENT	CFX / FLUENT	CFX / FLUENT	CFX / FLUENT



Prescribed motion



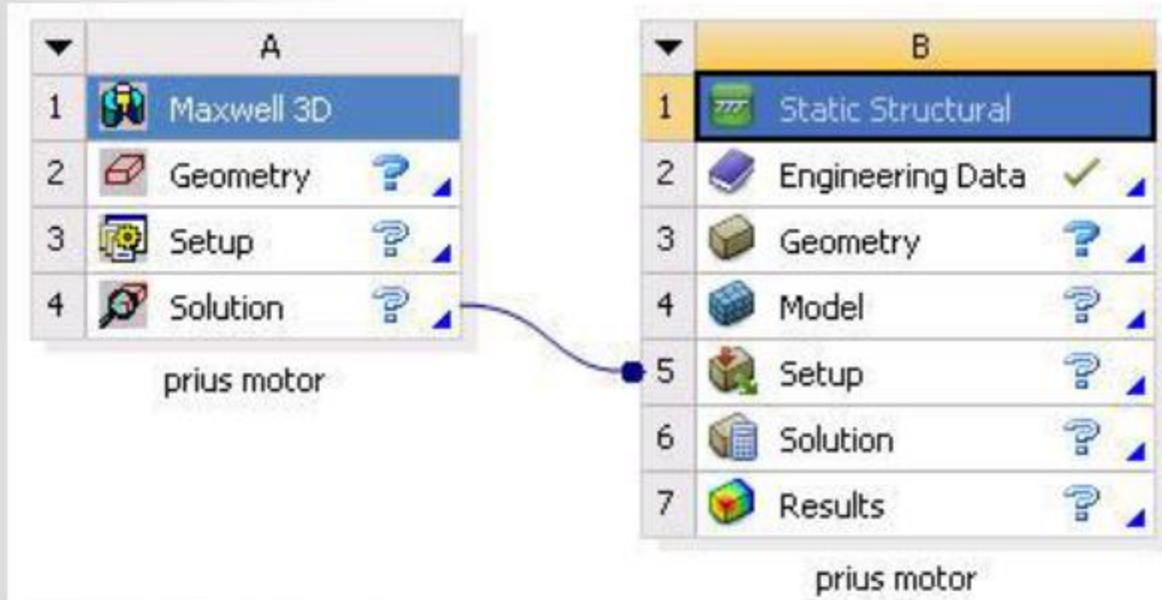
One-way, common practice

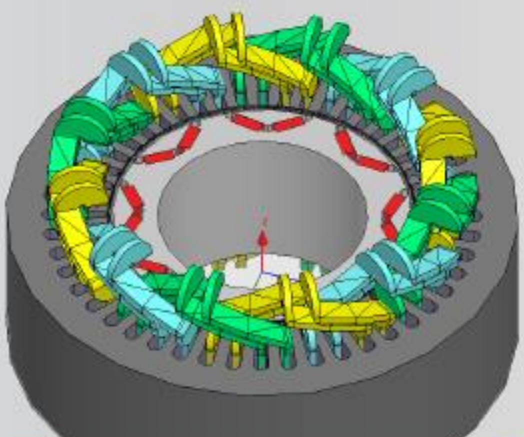


Two-way, high fidelity

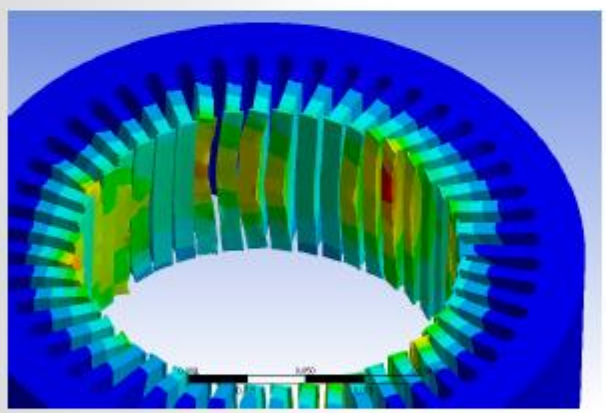
Electromagnetics and Structural Coupling

- **Maxwell provides volume/surface forces to ANSYS Mechanical**

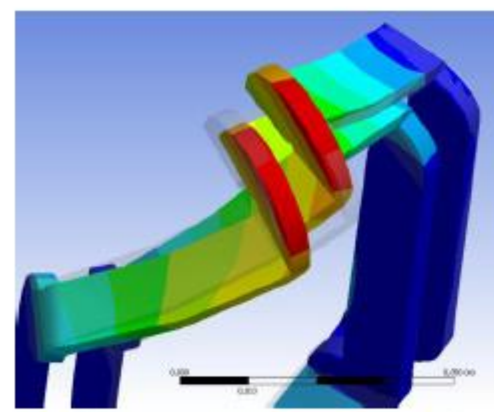




Electromagnetic force density from Maxwell is used as load in Mechanical

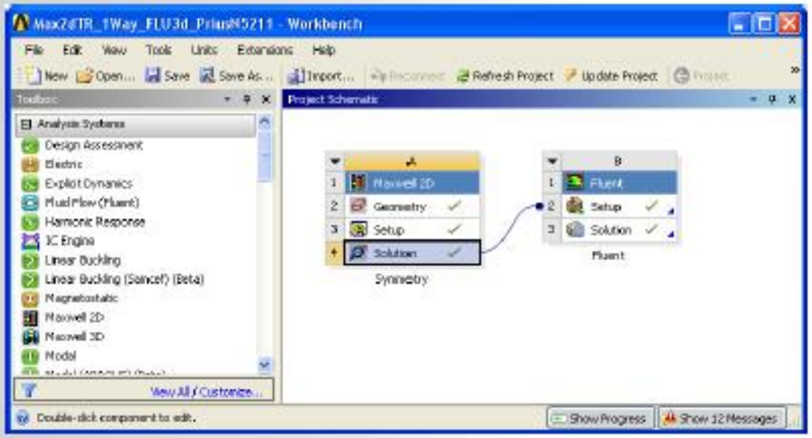


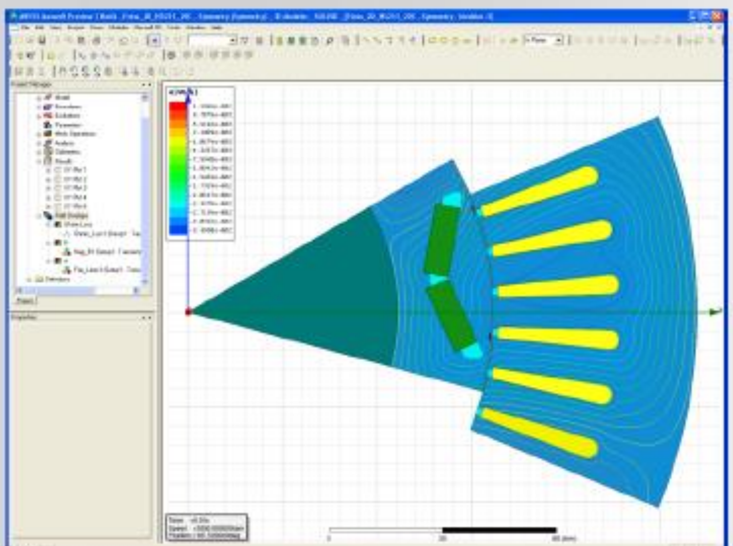
Deformation of the stator



Deformation of coils

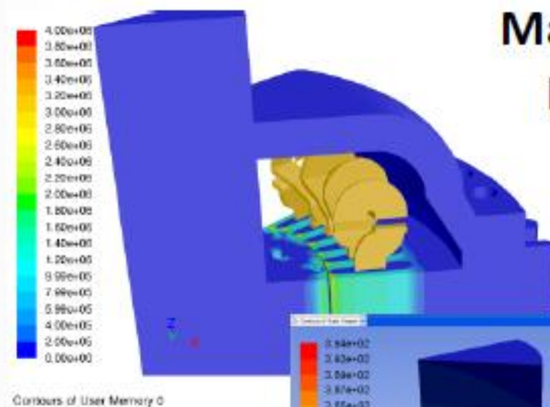
Electromagnetics and Fluid Coupling

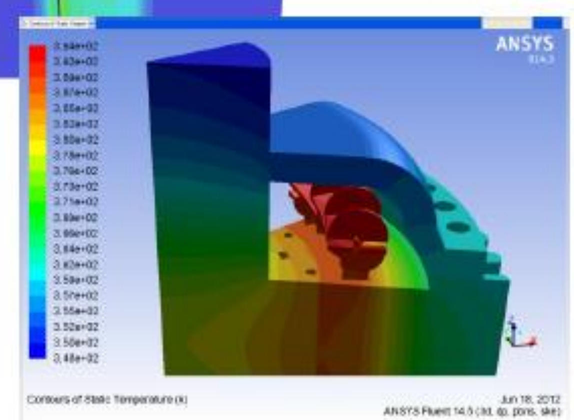




Electromagnetics

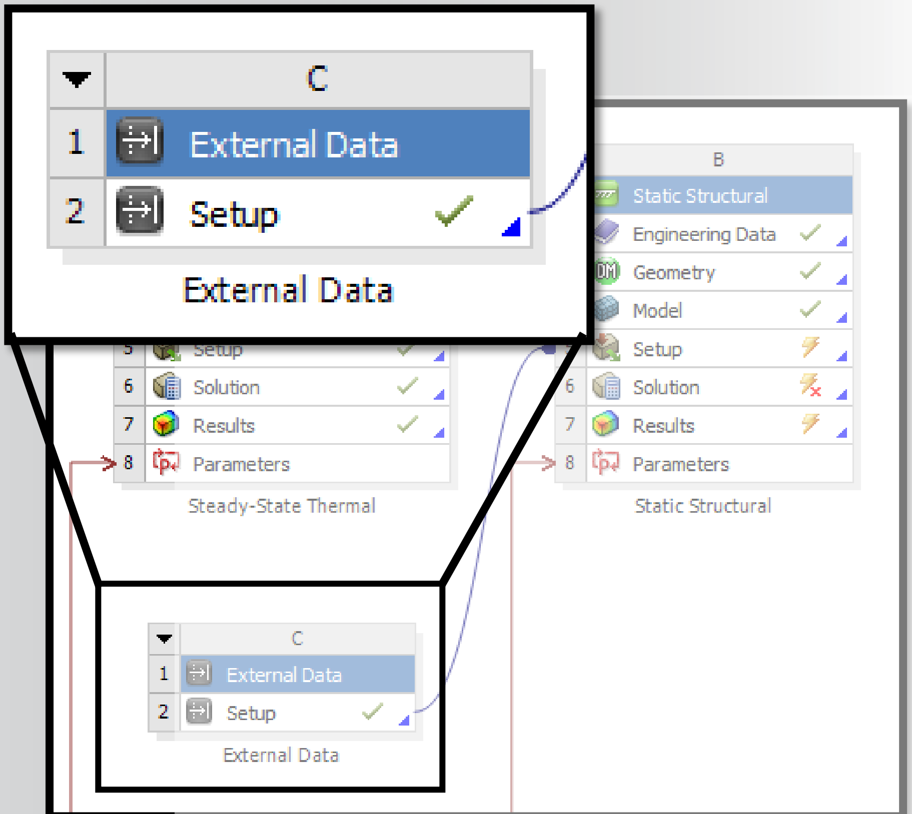
Mapped Loss

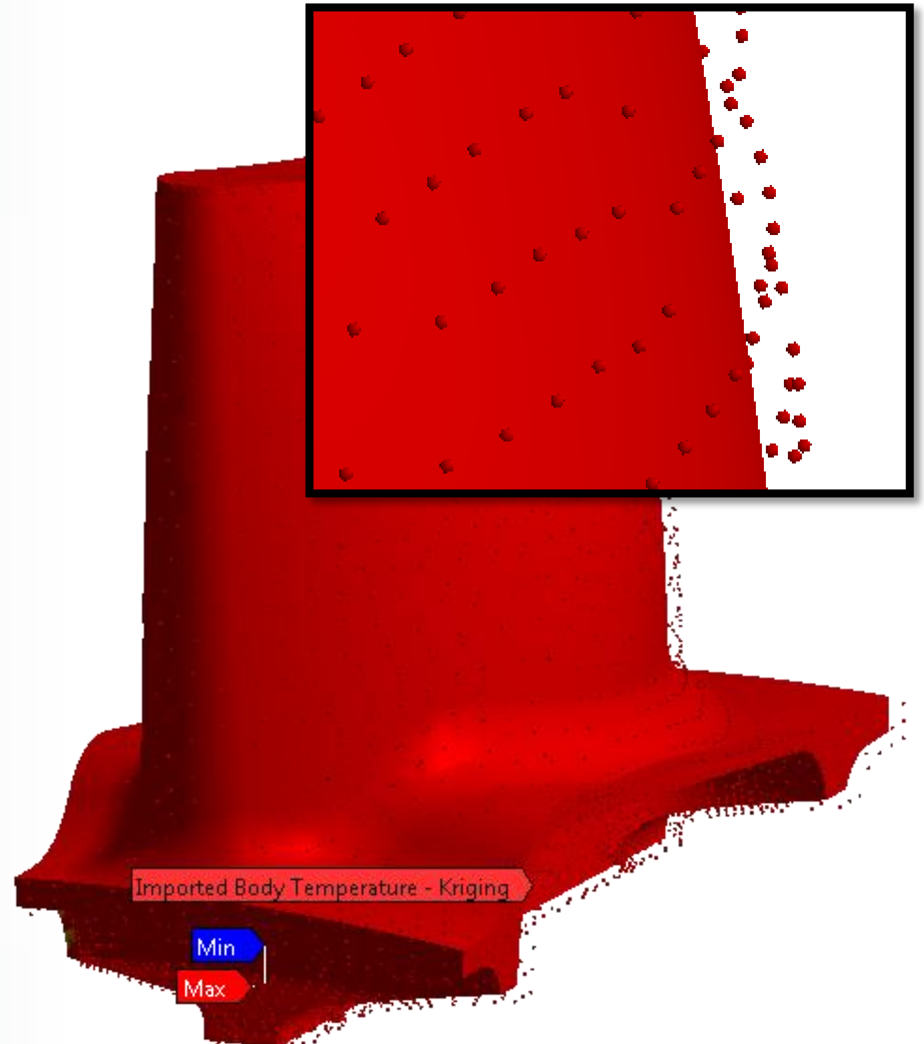




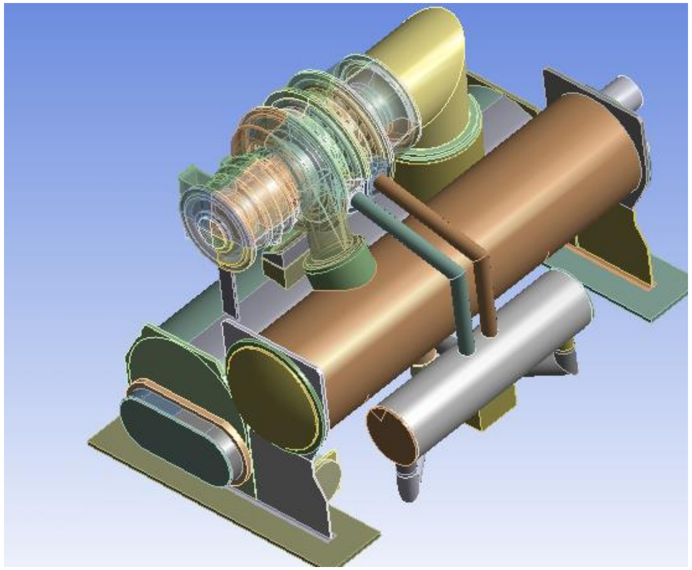
Temperature

In Addition To Multiphysics, Data Mapping Can Apply Measured/Experimental Results To Complex Surfaces



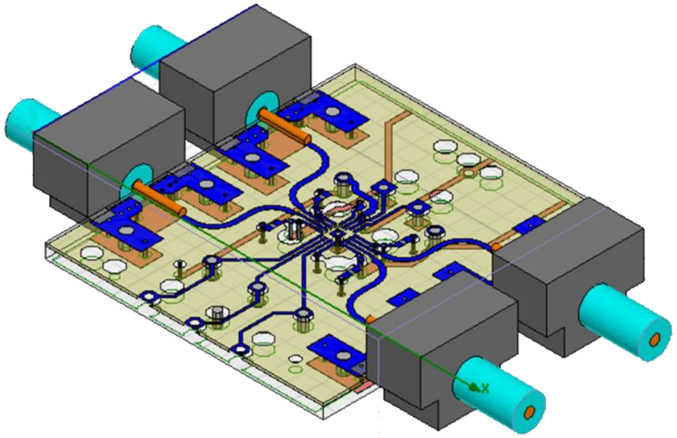


Connects to Design Environment in Multiple Ways



File Reader

- PTC: Creo Parametric
- Siemens: NX, Solid Edge
- Dassault: CATIA, Solidworks
- Autodesk: Inventor, AutoCAD
- Cadence: Allegro, APD, Virtuoso
- Synopsys: Encore
- Zuken: CR5000
- Mentor: Boardstation, Expedition, PADS
- Neutral Files: Parasolid, SAT, STEP, IGES, ODB++

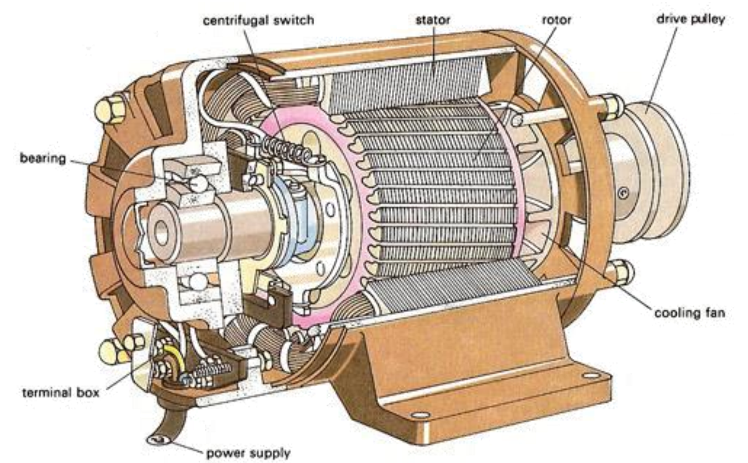


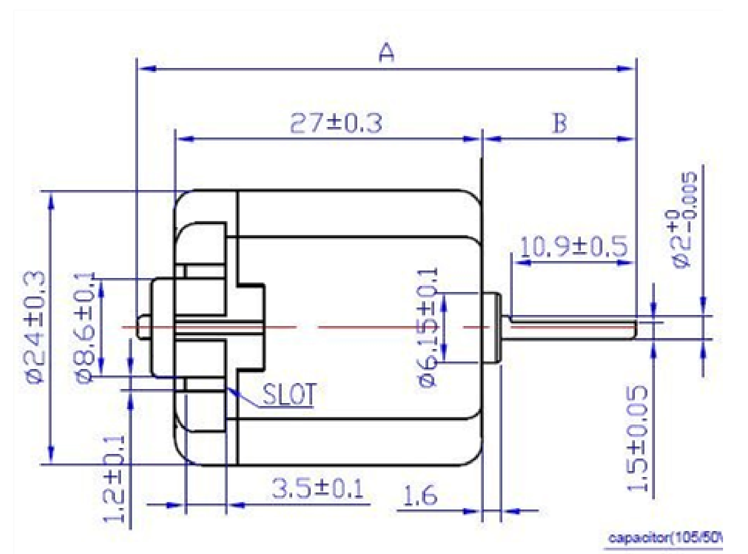
Bidirectional Associative Link

- PTC: Creo Parametric, Creo Direct Modeling
- Siemens: NX, Solid Edge
- Dassault: CATIA, Solidworks
- Autodesk: Inventor

Parametric Simulation

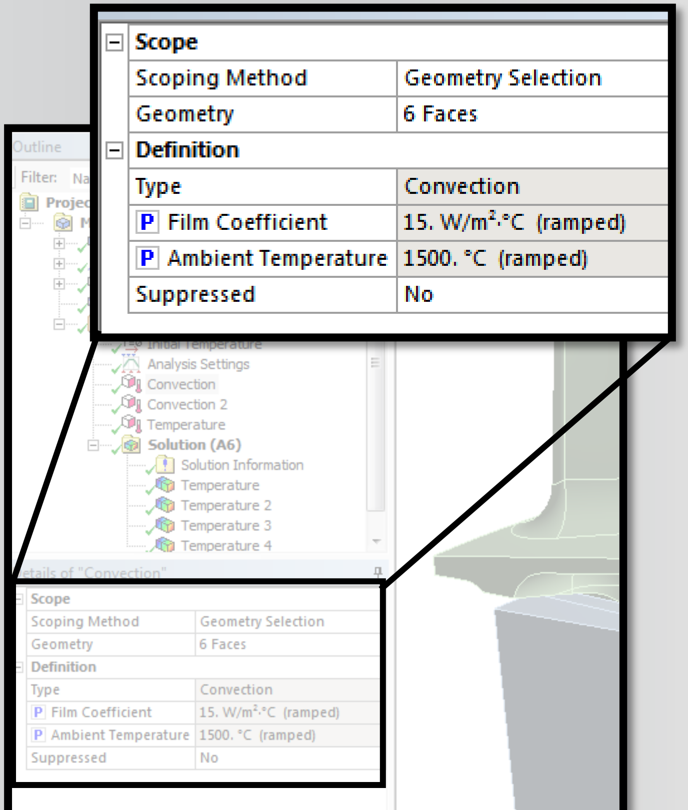
- **Today's Engineering Applications Create Parameters in Many forms:**
  - **Dimensions and Instances**
  - **Attributes and Names**
  - **Properties**
- **Parametric capabilities enable rapid updates of models for new designs and requirements scenarios.**



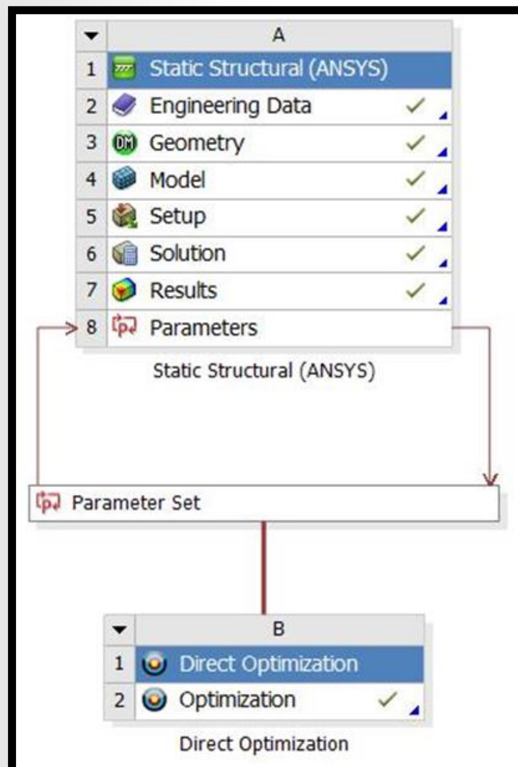


ANSYS Workbench Makes Parametric Simulation Part of a Standard Workflow

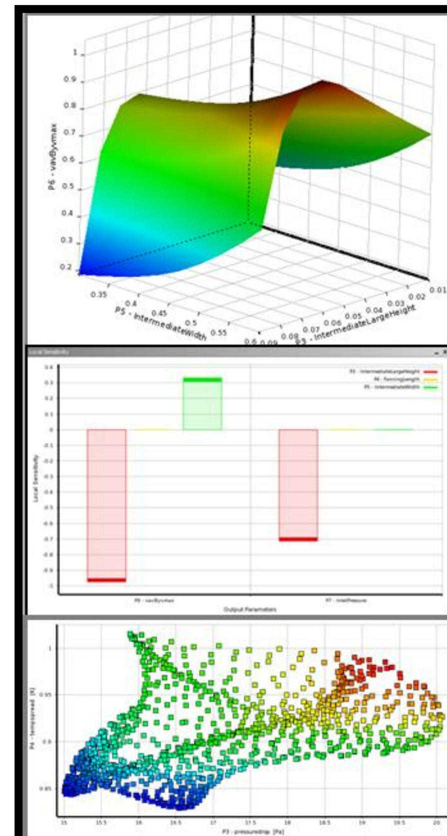
User Select Parameters



Workbench Collects Parameters from All Project Applications



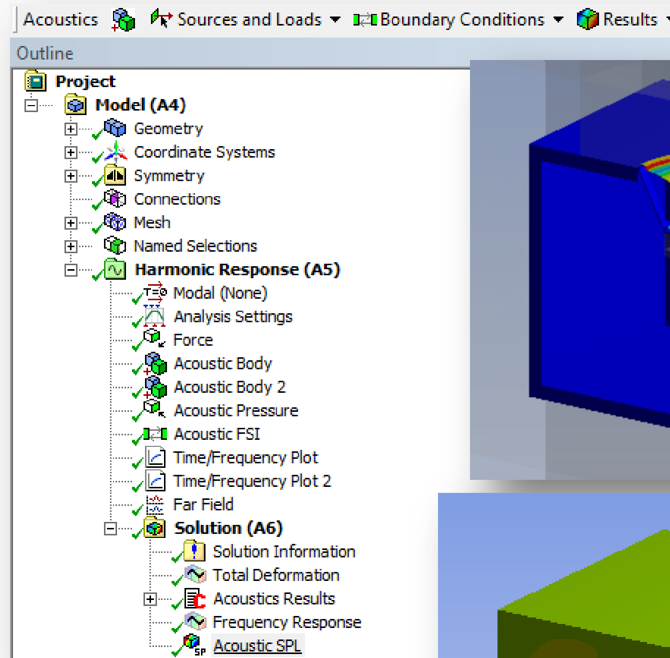
Results that Evaluate Trade-off Require Just a Few More Mouse Clicks

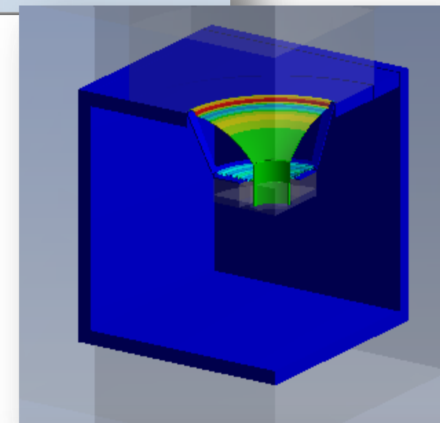


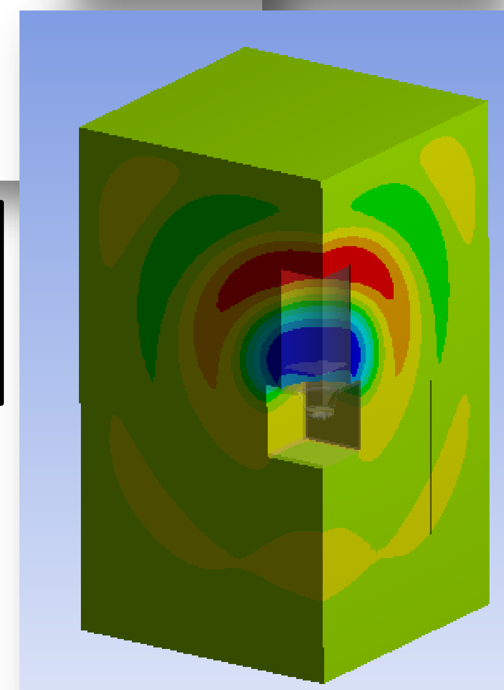
November 25, 2013

Highly Adaptable and Customizable

- **<u>ANSYS Customization Suite</u>**
- Expose custom calculations and results in a native environment
- Attractive solution for automation and democratization of simulation
- Available at 14.5



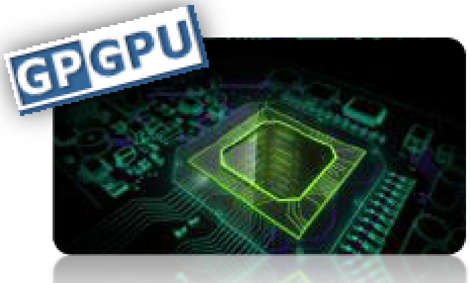


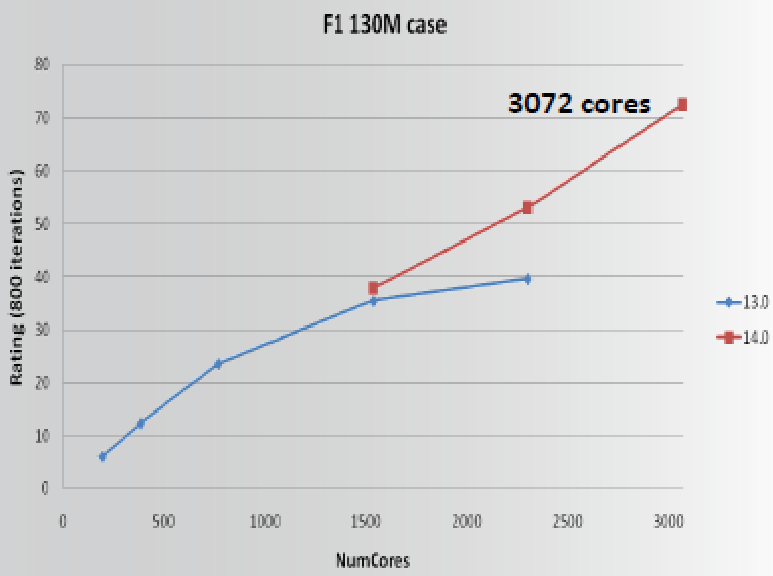


Example: Exposing MAPDL acoustics functionality in Mechanical

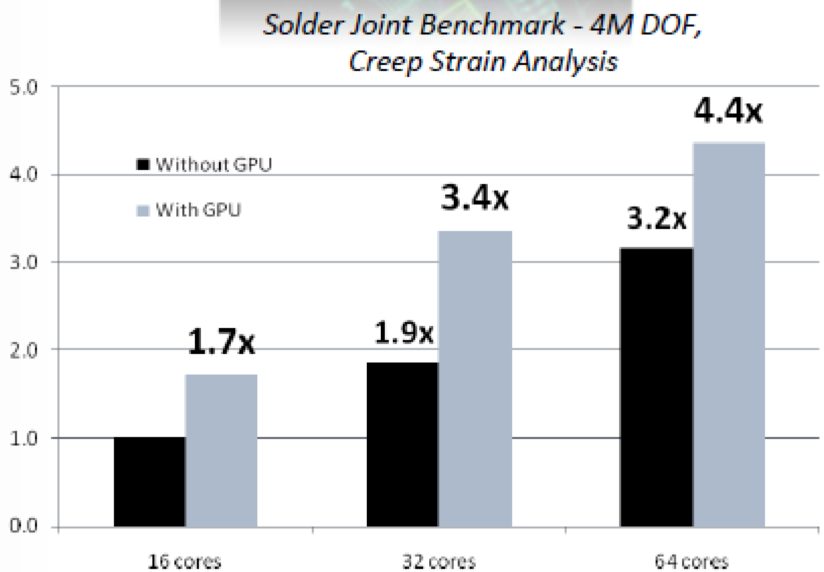
ANSYS HPC

- **High Performance Computing (HPC) adds tremendous value to simulations**
  - Enabling insight
  - Enabling productivity



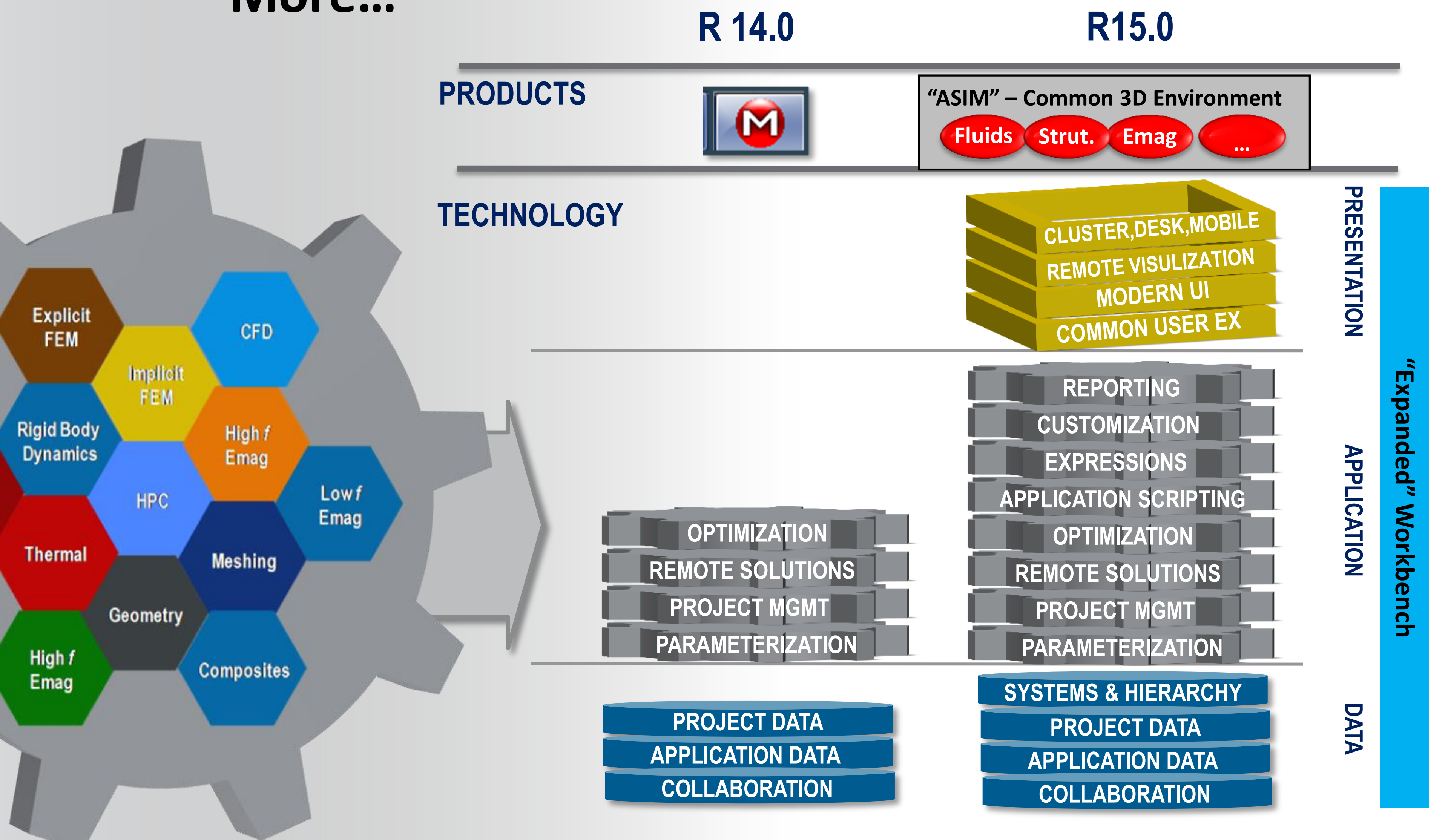


FLUENT Solver



Mechanical Solver

Architecture Provides a Unique Solution to Connect Solver Data and More…



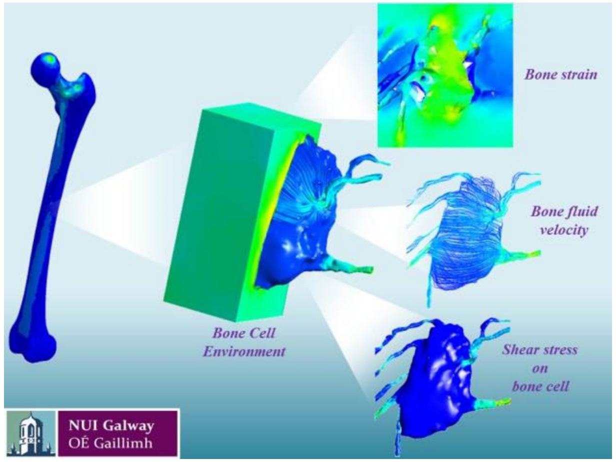
ANSYS Academic - examples

Problem

Due to the complexities of bone cells, the variables affecting bone growth are not easily identified and understood.

Solution

The Bone Mechanobiology Group at the National University of Ireland Galway uses ANSYS Multiphysics to provide a deeper understanding of the mechanics of bone cells. The new data provided from simulations allowed prediction of the stress and strain signals that simulate bone growth, as well as developing the next generation of cures for bone diseases.



"Our ANSYS simulations are now being used to inform the next generation of cures for bone diseases, such as osteoporosis."

Biomechanics Research Centre (BMEC), National University of Ireland Galway

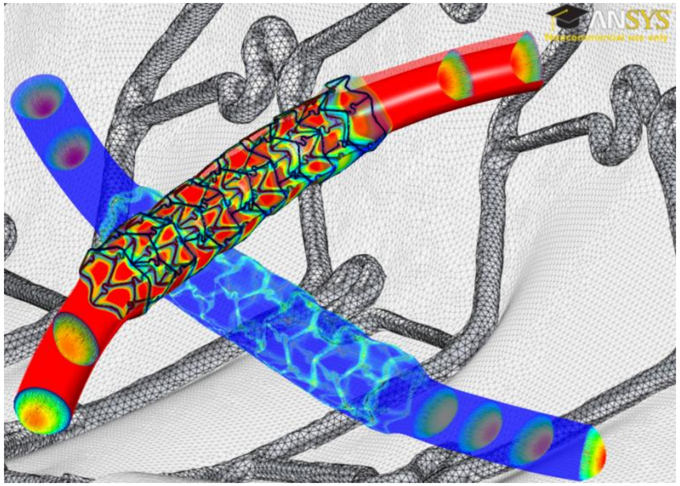
ANSYS

ANSYS Academic - examples

SVS FEM Your partner in computing

Problem

Patients who receive a cardiovascular stent may sometimes have post-treatment complications. Developments in healthcare engineering have the potential to decrease risks of post-operative disease and improve overall success rate.



Solution

University of Pittsburgh students used ANSYS CFD tools to simulate and analyze the complex interactions between the deployment of a cardiovascular stent, the perturbations of blood flow, altered mass transfer and the risk of post-operative diseases.

"Because of the flexibility of ANSYS tools, developing and using computational models covering a wide range of problems no longer requires considerable efforts or highly trained personnel."

University of Pittsburgh

ANSYS Academic - examples

Problem

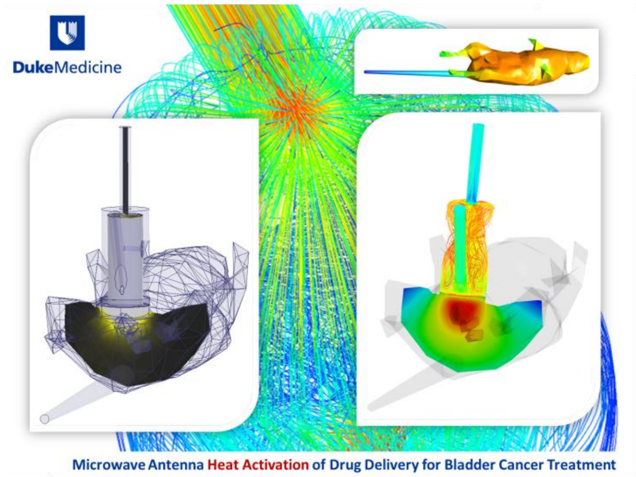
Discover a method to treat and eliminate cancer.

Solution

A group at Duke University is working towards a non-invasive approach to treat bladder cancer with the help of ANSYS HFSS and Fluent.

Result

They have developed a heated antenna that was able to deliver chemotherapeutics directly to the bladder without burning the skin. Currently, optimization of non-invasive heating tests on human bladders is ongoing.



By integrating ANSYS simulations at an early stage, it was possible to optimize the heat applicator without repetitive experimental testing. In the end, just a single prototype was needed and it performed as expected, resulting in a design efficiency of 100%.

Duke University,
Hyperthermia Group

ANSYS Academic - examples

Problem

Understand blood flow patterns before and after surgery to show whether a surgery has been successful.

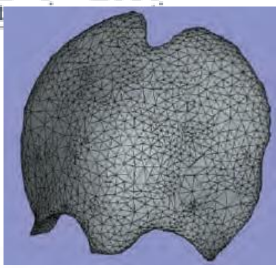
Solution

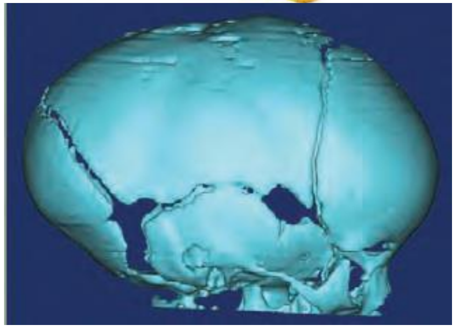
A research group at Linkoping University used ANSYS CFX to accurately capture the turbulent scales of blood flow through an aortic coarctation before and after surgery. They determined that a decrease in turbulent levels after the surgery which meant that the surgery was successful and that the workload of the heart decreased. Both quantitative and qualitative results agreed very well with MRI measurements.

The simulations provided an unprecedented detail of the flow, which is useful on both a clinical level when planning interventions and treatments, and on an engineering level in terms of understanding the flow features and transition to turbulence.

Linkoping University

ANSYS Academic - examples





Problem

As a newborn's brain grows, the skull must expand rapidly to accommodate it. In some cases, the bone plates of the infant's skull close before this growth is complete which can lead to developmental difficulties. Surgeries (osteotomies) are required but surgeons need to trade off between more cuts guaranteeing proper brain expansion and fewer cuts which mean less risk to the patient.

Solution

Use ANSYS structural mechanics software to help determine the placement and number of cuts required during surgery,

Result

Without simulation, doctors probably would have performed unnecessary additional osteotomies to ensure there was enough room for brain expansion

The simulation results helped the surgeon to prepare better for the operation;
the procedure itself was much faster, and it was easier on the infant because of the reduced number of osteotomies. Operations were successful on each of 20 patients, and the children are all doing well.

Medical University of Silesia

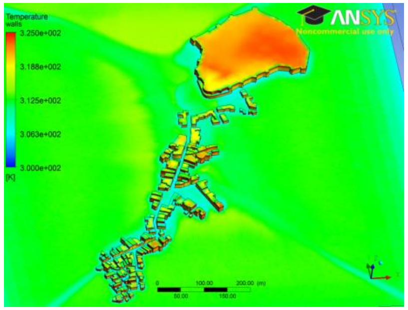
ANSYS Academic - examples

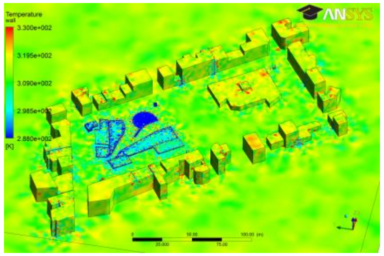
Problem

In heavily populated cities in Greece, using certain building materials and techniques in wall construction can help improve human thermal comfort.

Solution

Using ANSYS CFX to test various building materials, Democritus University of Thrace proposed that materials such as green roofs and water surfaces should be considered.





"ANSYS CFX solves coupled environmental and thermal flows even for large scale geometries where conventional models fail to succeed."

Democritus University of Thrace

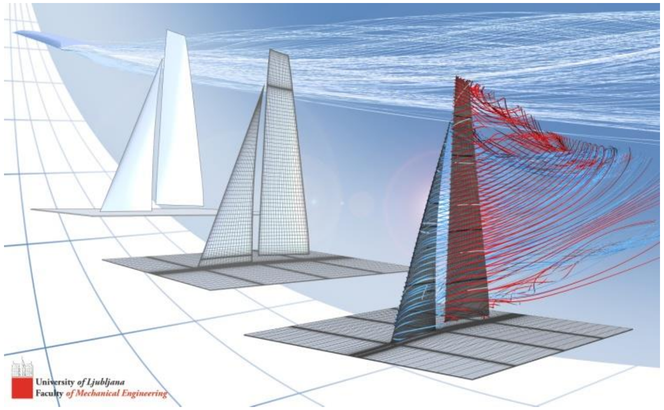
ANSYS Academic - examples

Problem

Racing yachts can move faster when sail design contributes to reduced drag in upwind conditions.

Solution

The University of Ljubljana studied pressure distribution on upwind sails using ANSYS CFX, applying turbulence models to compare computed and experimental results.



"Simulation enabled the team to collect data for reducing mainsail drag. By shortening analysis time, we could study more design variables, which led to an optimized design with maximum performance."

Laboratory for Aeronautics
University of Ljubljana

ANSYS Academic - examples

Problem

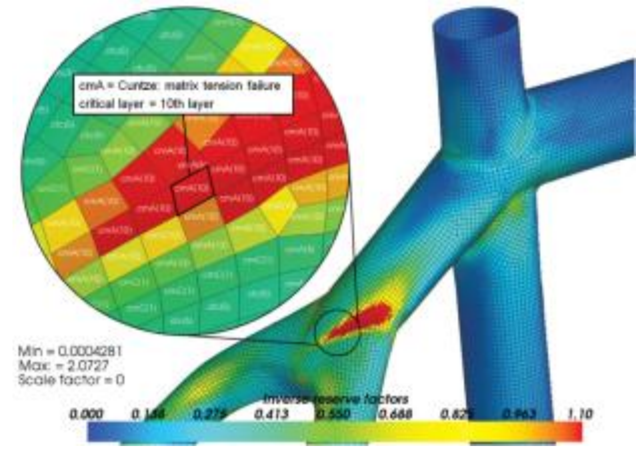
To successfully identify the stresses for a carbon-fiber–reinforced mountain bike frame for GHOST Bikes GmbH.

Solution

Used ANSYS Composite PrepPost software to simplify analysis of the carbon-fiber reinforced polymer (CFRP) bicycle components.

Result

- Stiffness and resistance characteristics of the bicycle components met design requirements including European standards.
- ANSYS Composite PrePost within the ANSYS Workbench environment significantly improved design efficiency.
- Compared to typical trial-and-error development methods used in the bicycle industry, the number of cost- and time-intensive physical prototypes was greatly reduced.



ANSYS Composite PrepPost software, integrated within the ANSYS Workbench environment, takes advantage of outstanding features and solver technologies from ANSYS. This technology substantially simplifies analysis of CFRP structures using innovative modeling and analysis capabilities.

Chemnitz University

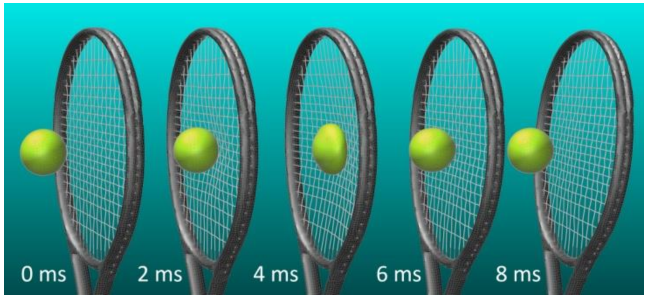
ANSYS Academic - examples

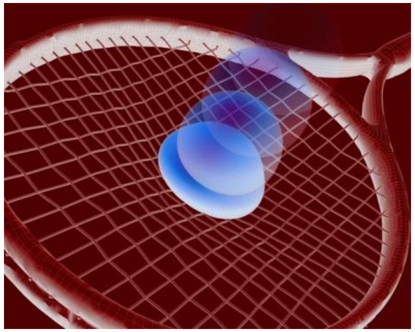
Problem

The tennis equipment market is highly competitive, so manufacturers must continually innovate, producing advanced equipment within very short time frames.

Solution

Sheffield Hallam University used ANSYS LS-DYNA to simulate a tennis ball impacting a racket. The model can be used as an engineering tool to accurately predict the effect of changing specific parameters, including structural rigidity or shot style.





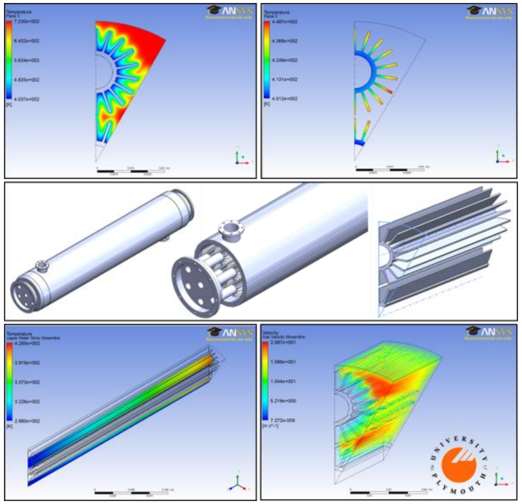
"The simulation model created with ANSYS software will reduce the time and cost of the R&D process. It can also help to develop tennis equipment that is specifically designed for the end user."

Sheffield Hallam University

ANSYS Academic - examples

Problem

Plymouth University students produced a concept design for a heat exchanger capable of heating domestic water to 40˚C from boat engine exhaust gases with minimal pressure losses and power requirements.



Solution

Students applied ANSYS DesignModeler, ANSYS Workbench and ANSYS CFD, which predicted water outlet temperature within 8 percent of theoretical values, providing time- and cost-effective analysis for the client.

"Through simulation, potential design improvements highlighted at an early state to improve heat exchanger performance."

University of Plymouth

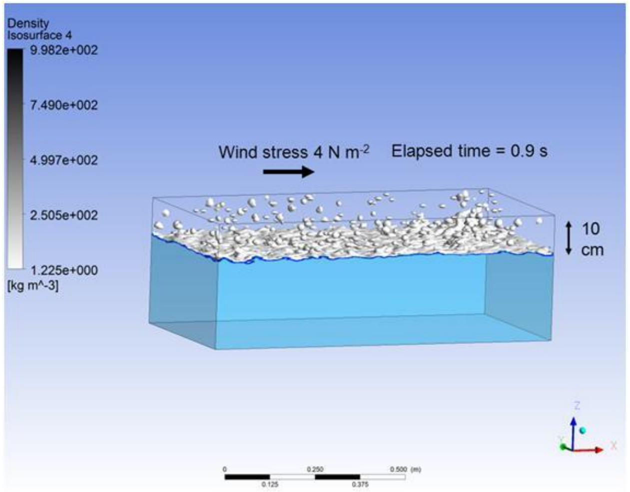
ANSYS Academic - examples

Problem

For hurricane prediction models, unresolved physics, especially near the air-sea interface, are the weakest components.

Solution

Nova Southeastern University used ANSYS Fluent to simulate the process of disruption of the air–sea interface under hurricane conditions. The results of the study justify the extension of existing theory of wind–wave generation to hurricane conditions by including the effect of two-phase environment on ultragravity waves.



This project has been supported by the National Ocean Partnership Program (via NSU subcontract to the University of Rhode Island) and by the Gulf of Mexico Research Initiative (via NSU subcontract to the University of Miami/CARTHE).

"Simulation has played a significant role in the dramatic improvements that HEAD has achieved in the performance and durability of its tennis rackets over the past decade."

Oceanographic Center
Nova Southeastern University

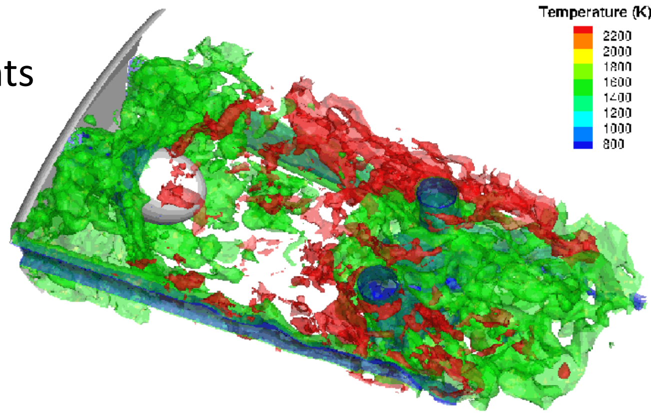
ANSYS

ANSYS Academic - examples

SVS FEM
Your partner in computing

Problem

Gas turbine combustors produce turbulent combustion creating a strong reliance on experimental measurements and expensive setup costs.



Solution

Cranfield University used ANSYS Fluent to simulate turbulent combustion in gas turbine combustors to provide a better understanding of the complex flow phenomena involved and predict quantities such as velocity, temperature, combustion products and pollutants with high fidelity and at a relatively low cost.

"The results obtained from ANSYS Fluent were used to calibrate simpler in-house combustion performance and emission models."

Cranfield University

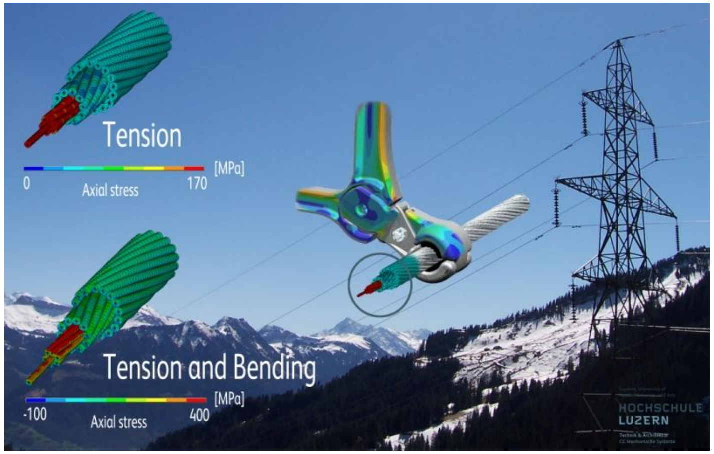
ANSYS Academic - examples

Problem

As electrical energy requirements increase, companies look at new options to provide energy to customers through improved infrastructure.

Solution

Researchers at Lucerne University used ANSYS Mechanical to test, improve and develop more reliable and practical common overhead lines.



By using ANSYS simulations, researchers were able to better understand the complexity of common overhead lines to improve the technology..

Lucerne University of Applied Sciences and Arts

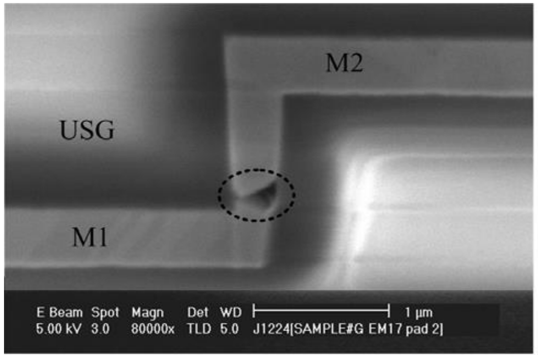
ANSYS Academic - examples

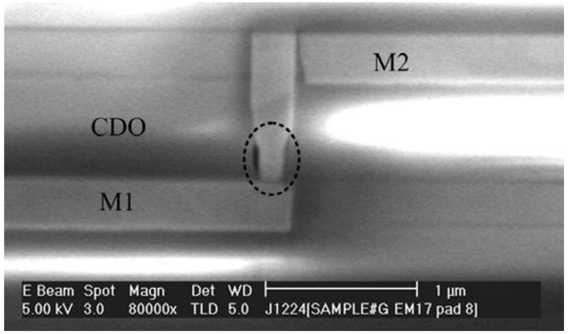
Problem

Interconnect reliability is crucial to the reliability of integrated circuit that consists of billions of transistors interconnected to form a circuit. Stress in the interconnect lines during fabrication can form voids and create an open circuit, decreasing the reliability of the integrated circuit.

Solution

Nanyang Technological University used ANSYS multiphysics modeling to understand why the voids are caused and can fine tune the fabrication process to prolong the lifetime of an integrated circuit.





Multiphysics modeling clearly explains the mechanism of the voiding.

Nanyang Technological University

ANSYS Academic - examples

Problem

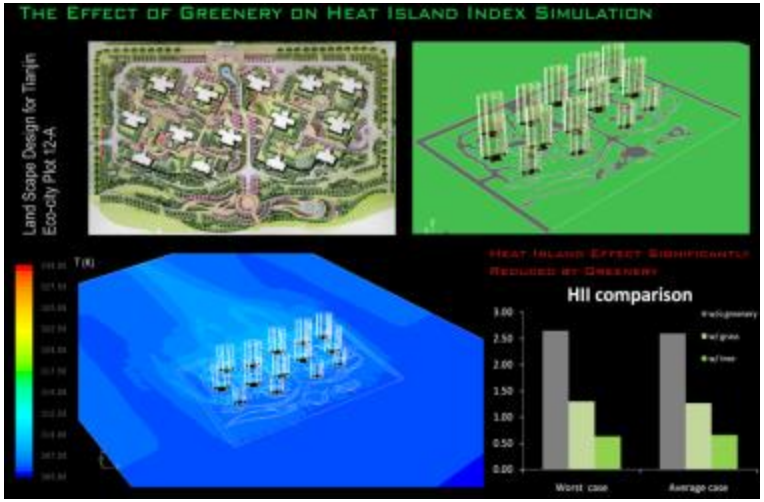
Study the Heat Island Index of a piece of land

Solution

The simulation, conducted in Fluent, took a 2-D mesh of a landscape design of a plot of residential buildings. The mesh was imported into Gambit and the whole plot was generated.

Result

The simulation showed the Heat Island Index was reduced by the greenery on the plot of land.



ANSYS Fluent and Gambit was used to study the Heat Island Index for a plot of land. The study concluded that greenery reduced the Index number.

Carnegie Mellon University

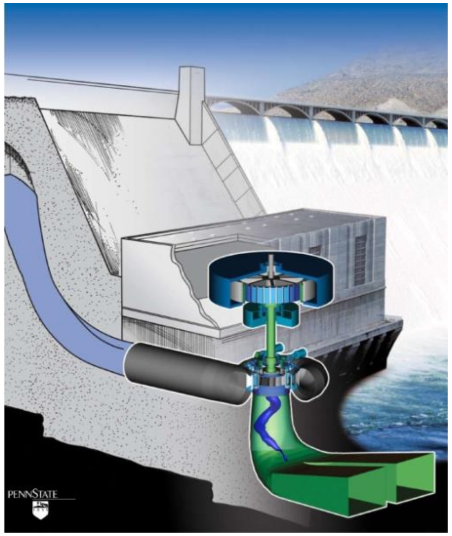
ANSYS Academic - examples

Problem

Hydro-electric plants must operate efficiently and economically. Unstable water flow produces a vortex rope that increases pressure fluctuations and reduces turbine efficiency.

Solution

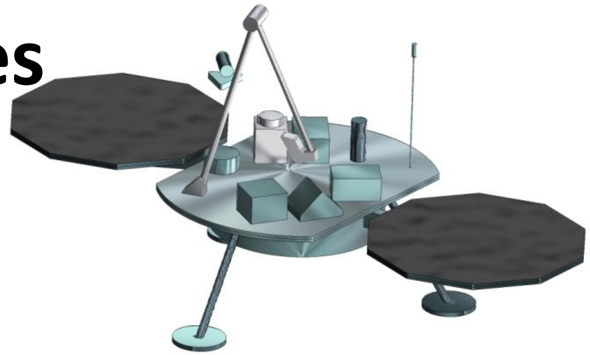
Using ANSYS Fluent, Pennsylvania State University analyzed the complex phenomena occurring in the draft tube. This is enabling the team to investigate control techniques to prevent vortex rope formation and to improve draft tube performance.



"By using simulation, detailed features of the flow can be studied that were impossible to capture using experiments."

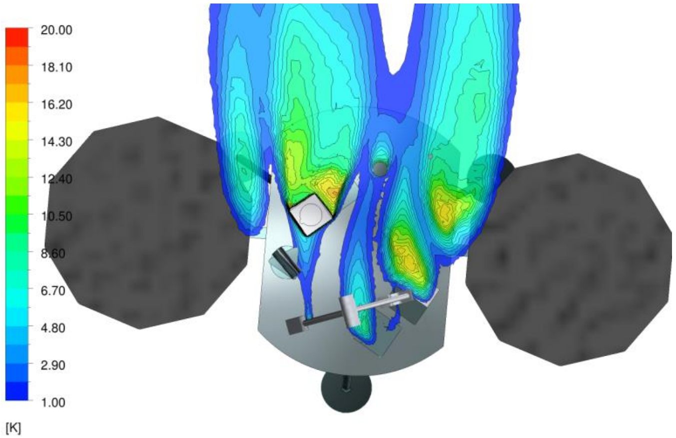
Penn State Hydropower Research Program

ANSYS Academic - examples



Problem

To calibrate the “weather station” for the Phoenix Mars Lander so that lander itself does not affect the readings. During the mission to quickly adjust the data to aid in the next day’s mission.



Solution

Calibrate instruments pre-flight using fluid dynamics then during the mission use CFD and HPC to provide rapid turnaround to guide data gathering for the next day’s mission.

Result

Data gathering proved successful in both phases and this gave time for additional simulations to be performed to help explain certain phenomena found in the raw data.

Parallel processing and the multi-domain scheme in ANSYS CFX combined with AMD’s multi-core architecture enabled simulations to be completed within the timeframe for decision making.

University of Alberta

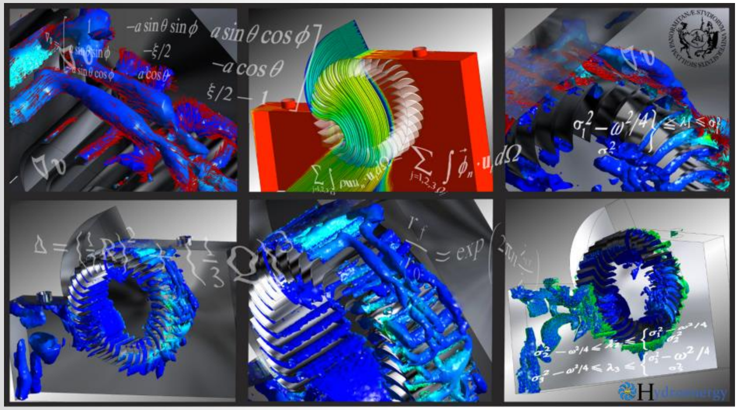
ANSYS Academic - examples

Problem

Researchers at the University of Palermo needed to design a new cross flow turbine.

Solution

ANSYS CFX was used in a two phase fluid (water and air) and a rotor-stator domain simulations. The efficiencies of the new turbine design was determined with the simulations.



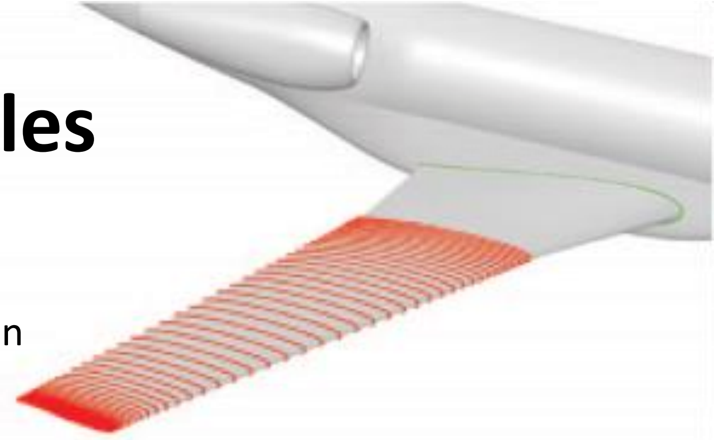
ANSYS simulations provided researchers the ability to design, test and improve the efficiency of the cross flow turbine prior to the prototype stage.

Università degli Studi di Palermo

ANSYS Academic - examples

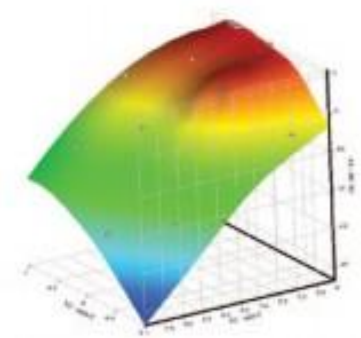
Problem

Wing design is normally tested one iteration at a time and can take many months to optimize a design.



Solution

Piaggio Aero Industries teamed with the University of Rome Tor Vergatta to design a new optimization method that generates a single mesh and morphs it to any new geometry using ANSYS DesignXplorer, ANSYS ICEM CFD and RBF Morph.



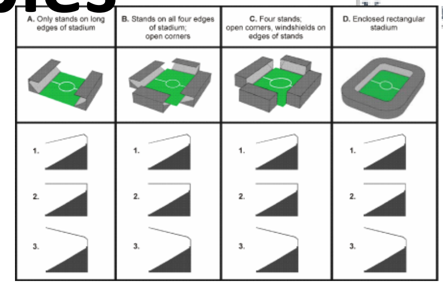
Result

Researchers were able to evaluate the robustness of the various wing designs to determine the ones that delivered consistent performance as design parameters were varied.

With ANSYS software, the design optimization took a couple of weeks, less than a tenth of the time required to optimize the design using conventional methods.

University of Rome Tor Vergata and Piaggio Aero Industries

ANSYS Academic - examples



Problem

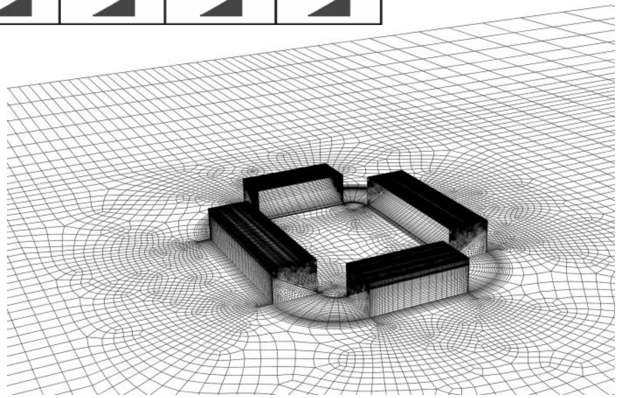
When designing a stadium, builders must consider many factors – two of the overall considerations are spectator comfort/safety and how the stadium could affect play.

Solution

Conduct 3-D studies of stadium designs using ANSYS CFD software to determine the ideal architectural designs to limit wind flow and wind-driven rain.

Result

Planning for these venues can now incorporate wind-flow patterns and wind-driven rain to optimize the stadium design, and include cost saving techniques.



Simulation results can be used to improve the design of stadiums as well as to diagnose and correct problems with existing stadiums – such as using special paint to protect seats that frequently get wet, to reduce maintenance costs.

Eindhoven University of Technology

Courtesy van Hooff & Blocken.

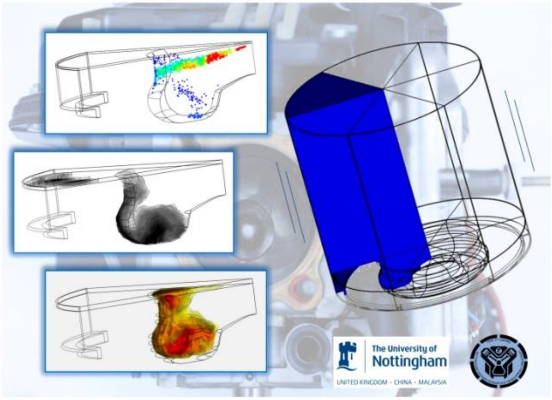
ANSYS Academic - examples

Problem

Students at the University of Nottingham discovered damage from an unknown source to the engine piston.

Solution

With the help of ANSYS Fluent, students discovered that soot deposition on the cylinder liner and entrainment into the engine's oil correlate to oil starvation and damage to the engine piston.



"With ANSYS Fluent, detailed visualization of complex diesel combustion shows the interdependence between the soot entrainment process and the in-cylinder gas motion, the location of combustion and evolution of the soot cloud."

University of Nottingham

ANSYS Academic - examples

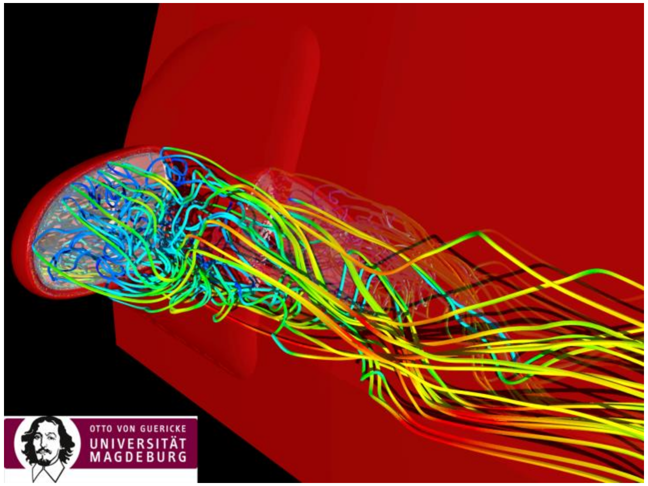
SVS FEM
Your partner in computing

Problem

As oil prices soar, consumers and governments are demanding improved vehicle gas efficiency. Even minor improvements can help manufacturers to improve mpg while meeting industry standards and requirements.

Solution

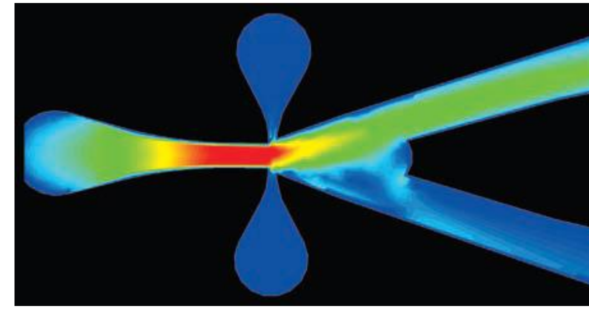
The University of Magdeburg used ANSYS CFD to determine unsteady turbulent flows around a car's side-view mirrors. The software's built-in models and user-defined functions — including consideration of a film flow on the surface of investigated geometry — enabled the detailed analysis.



"ANSYS high-quality mesh provides high accuracy and fast convergence compared to the fully unstructured mesh."

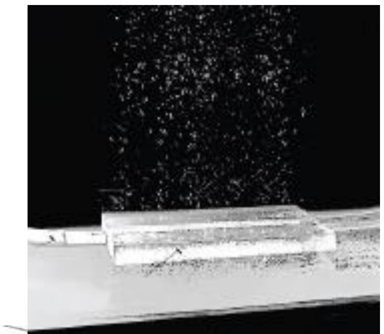
University of Magdeburg

ANSYS Academic - examples



Problem

Manufacture less expensive biofuels by introducing microbubbles of CO_2-rich gasses into the bioreactor.

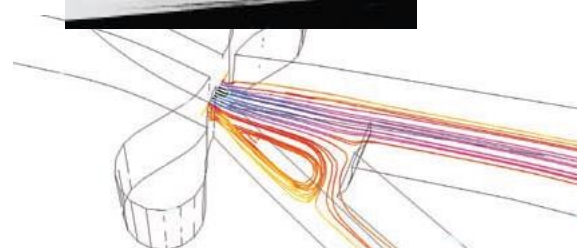


Solution

Use ANSYS Fluent to help develop an oscillator that delivers microbubbles of appropriate diameter to enhance algae growth.

Result

With the 50-fold increase in mass transfer rate afforded by this oscillator design, CO_2 dispersal is accelerated in the bioreactor and should enhance algae growth rate by a factor of 10. The experiments showed an 18 percent reduction in the energy required for bubble production compared to conventional fine bubbles.



The team predicts that engineering simulation will play an even greater role in the coming design of a commercial-scale bioreactor, when it will become critical to optimize all components of the design to minimize capital expenses.

University of Sheffield

CFD Simulations of 2.5 MW turbine using ANSYS CFX and OpenFOAM

Bastian Dose[1,2], Wided Medjroubi[3] and Bernhard Stoevesandt[2]
[1] University of Applied Science Kiel
[2] Fraunhofer IWES, Oldenburg
[3] ForWIND, Oldenburg

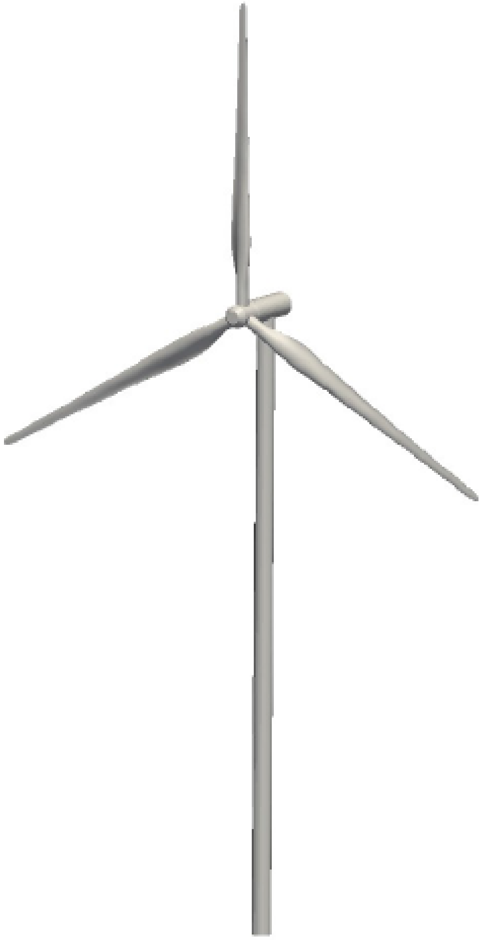
First Symposium on OpenFOAM in Wind Energy 2013, March 21th, Oldenburg

Outlook

- Complete wind turbine (incl. tower) was simulated
- Focus on comparison of ANSYS CFX and OpenFOAM
- Structured mesh generated in ANSYS ICEM CFD
- Steady-state and transient simulations

Turbine Data

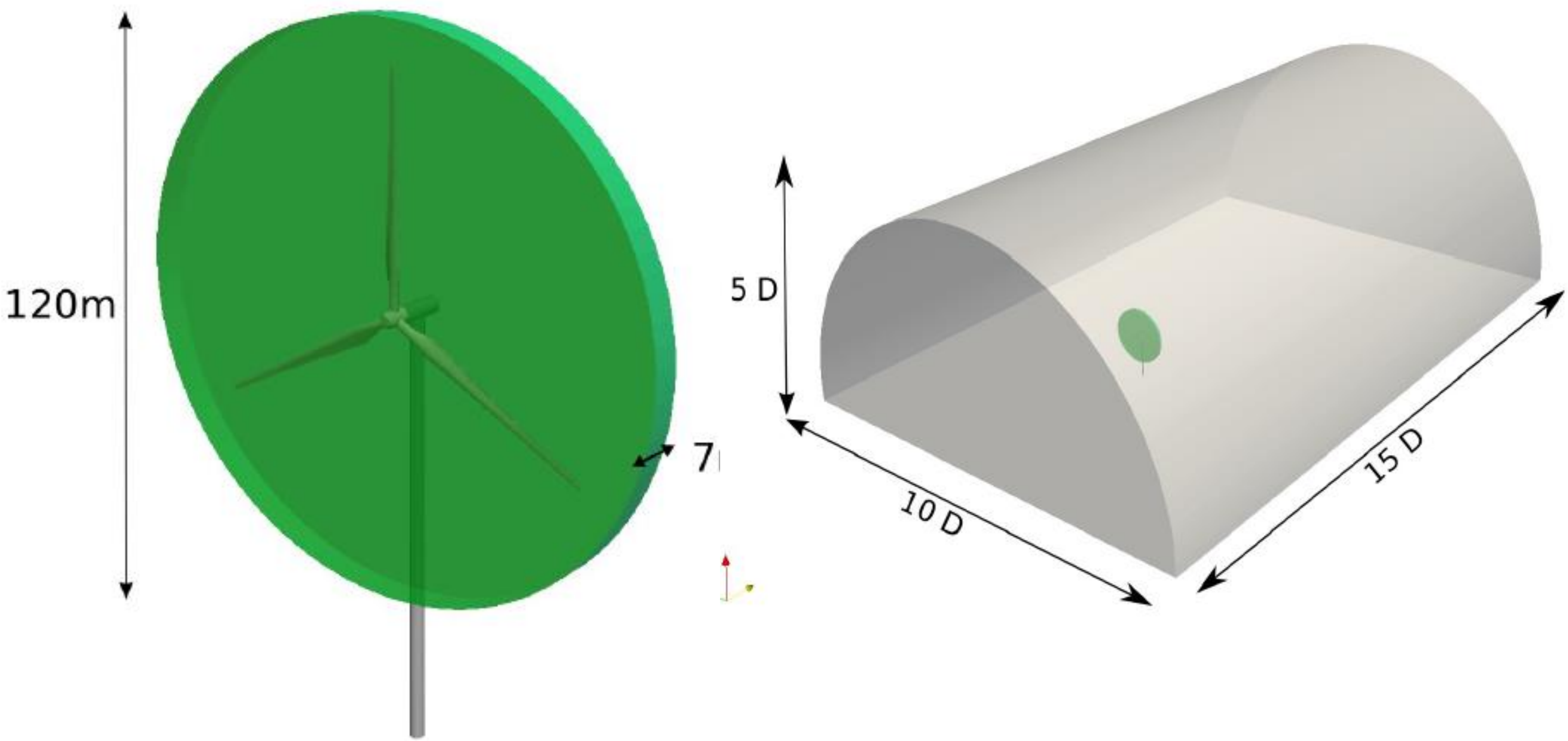
- Reference 2.5 MW wind turbine
- Designed by IWES
- Upwind configuration
- Rotor diameter: 100 m
- Hub height: 100 m
- Rated inflow velocity: 10.8 m/s
- Rated rotational speed: 13 rpm



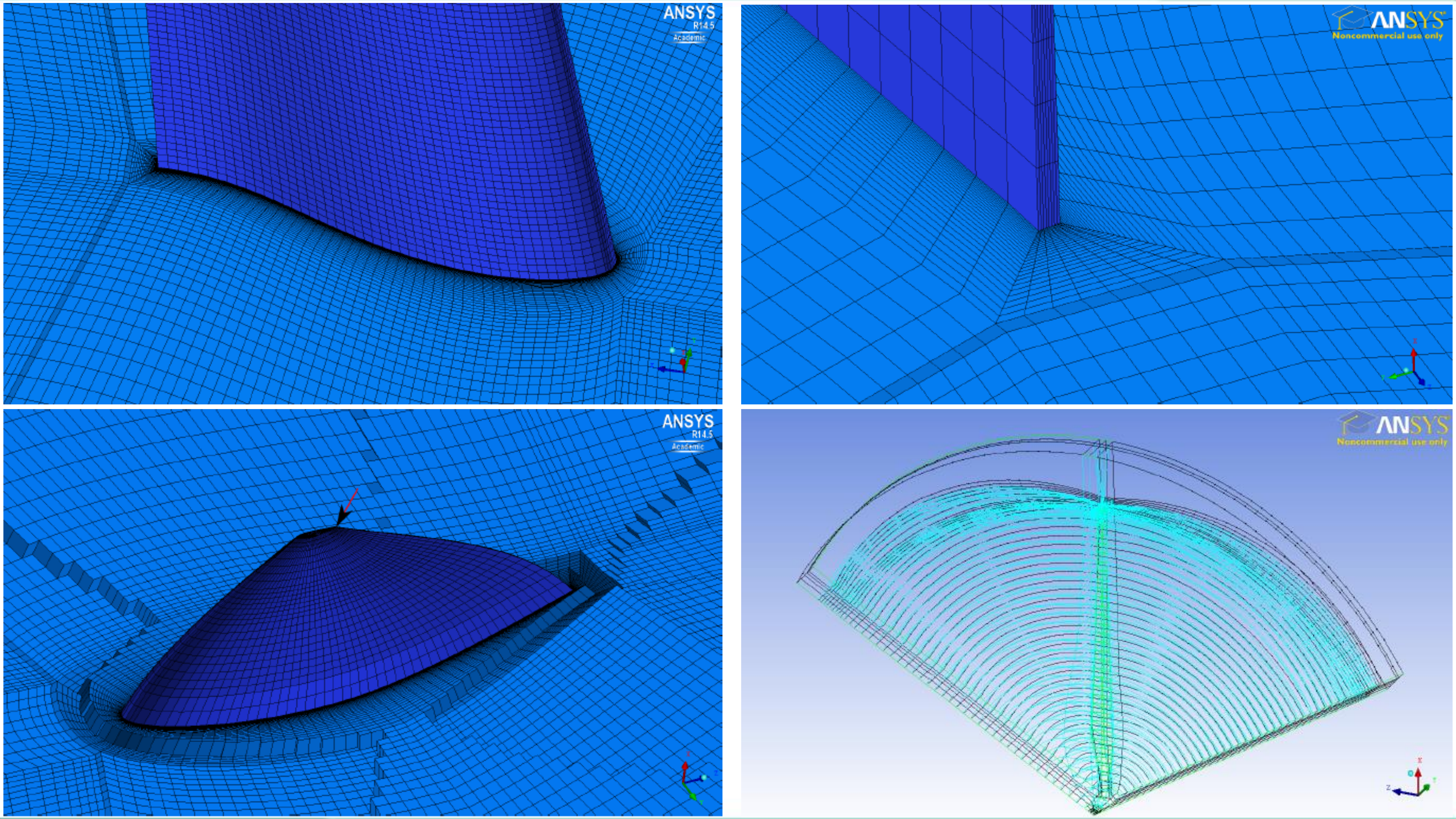
Meshing strategy

- Two separate structured meshes (rotor and far field)
- Both simulations use the same mesh
- Total cell count: 52 million (36 + 16)
- Mesh quality verified by checkMesh
- Y+ 200

Meshing strategy

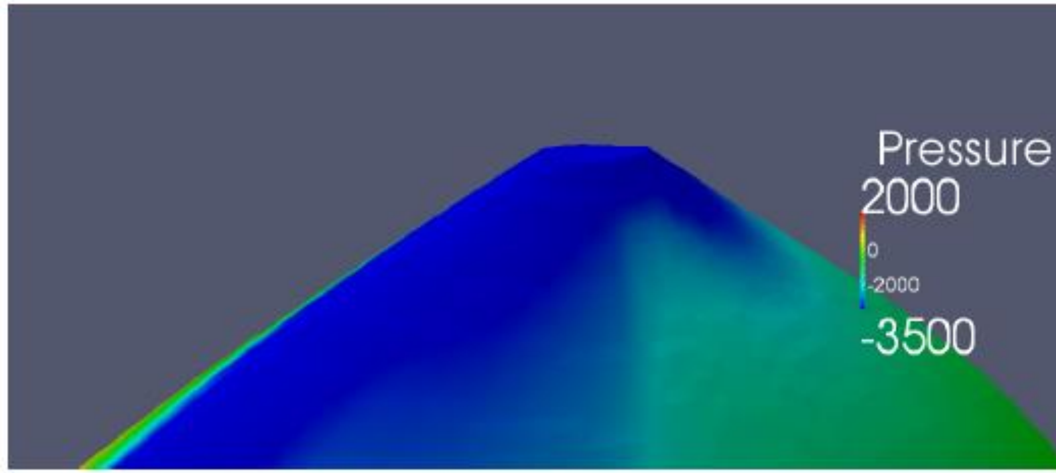


Meshing strategy

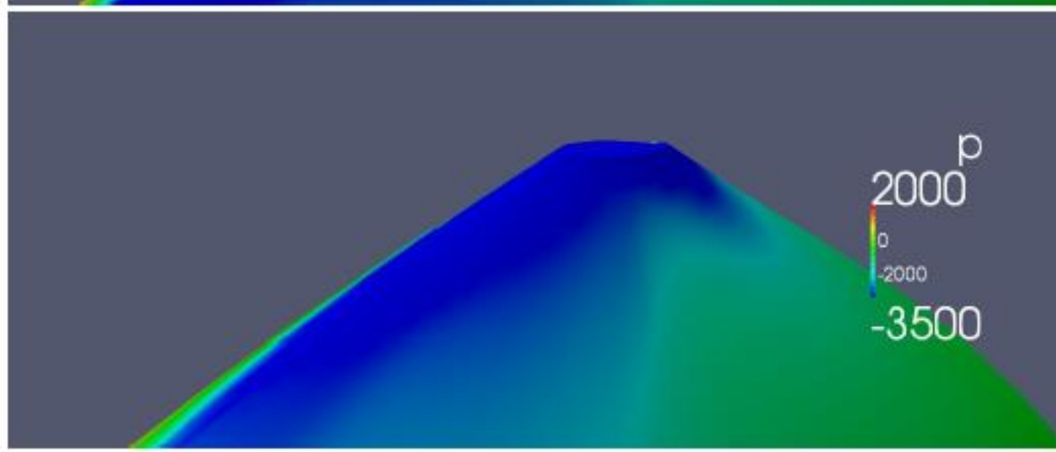


Preliminary Results

ANSYS CFX



OpenFOAM



Conclusion

- Structured meshes generated in ICEM can be used within OF
- Meshing approach used worked
- Similar results with both ANSYS CFX and OpenFOAM
- CFX faster (3-4x) and more stable

Realizace HPC výpočtů

- **RSM – Remote Solve Manager**

  ...implementace, uživatelské rozhraní, konfigurace, integrace plánovačů PBS/LSF, monitoring...

- **Specifická konfigurace samostatných komponent**

  ... Mechanical APDL, Fluent a CFX, aktivace GPU...

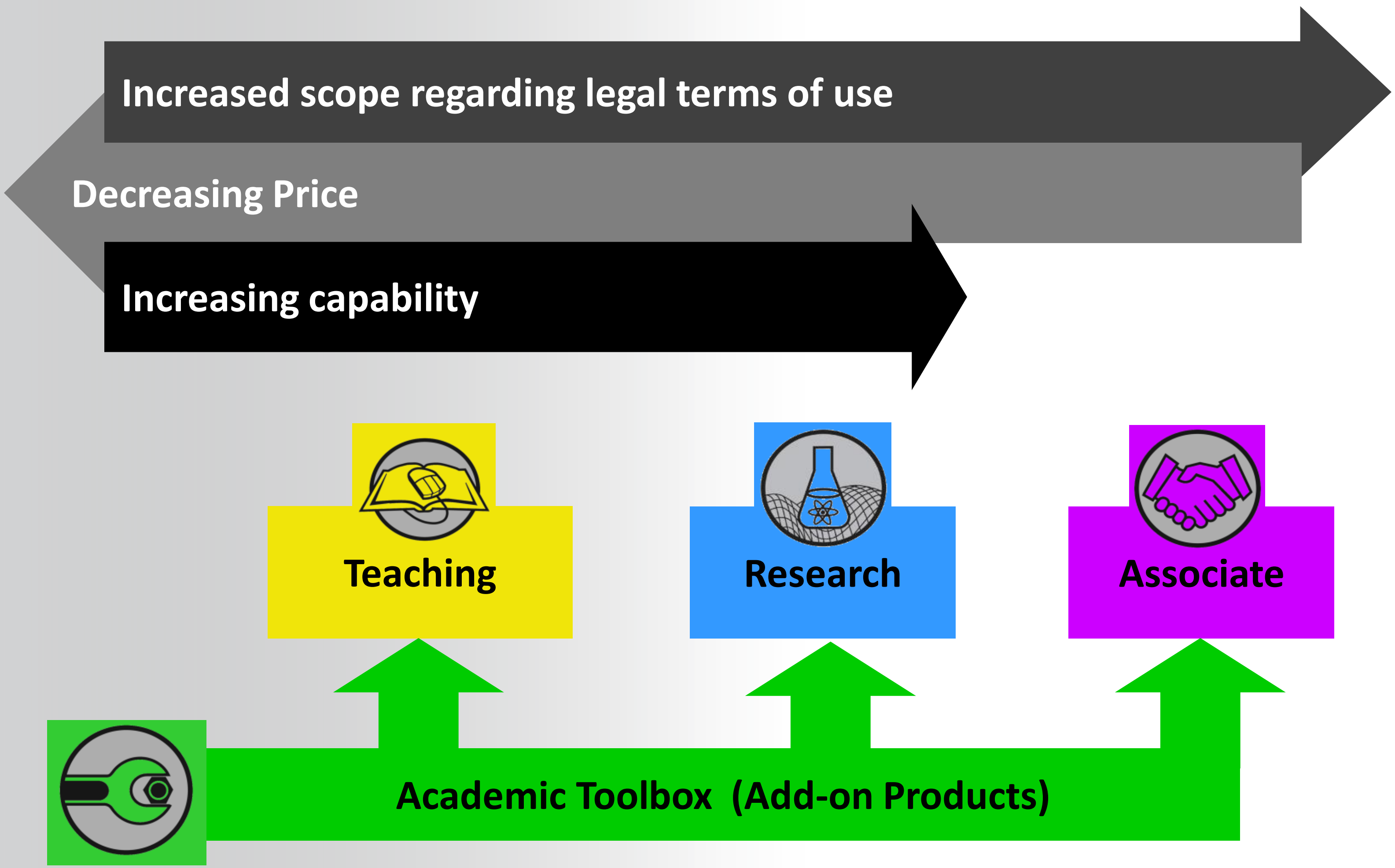
- **ANSYS Cloud**

  ...NICE DCV, EngineFrame, Vcollab, EKM...

14.5.7 ANSYS Structural & Fluid Dynamics Academic Portfolio

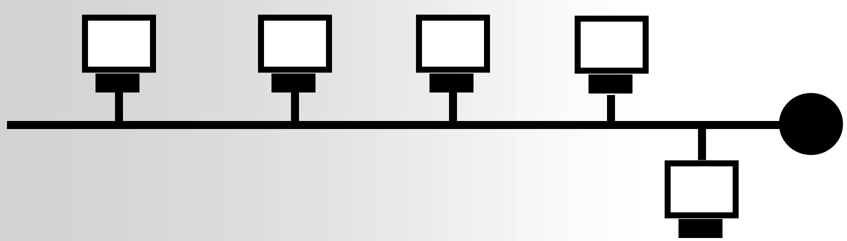
Product/Family	Task variants available
ACADEMIC ASSOCIATE	
ANSYS Academic Associate	1,5
ANSYS Academic Associate CFD	1,5
ANSYS Academic Associate HPC	1
ACADEMIC RESEARCH	
ANSYS Academic Research	1,5,25
ANSYS Academic Research Mechanical	1,5,25
ANSYS Academic Research CFD	1,5,25
ANSYS Academic Research Autodyn	1,5,25
ANSYS Academic Research Electronics Thermal	1,5,25
ANSYS Academic Research Offshore/Marine	1,5
ANSYS Academic Research Polyflow	1,5,25
ANSYS Academic Research HPC	1
ANSYS Academic Research LS-DYNA	25
ANSYS Academic Research LS-DYNA HPC	1
ACADEMIC TEACHING	
ANSYS Academic Teaching Advanced	5,25,50
ANSYS Academic Teaching Introductory	5,25,50
ANSYS Academic Teaching Mechanical	5,25,50
ANSYS Academic Teaching CFD	5,25,50
ACADEMIC TOOLBOX	
ANSYS Academic Meshing Tools	1,5,25
ANSYS Academic CFD Turbo Tools	1,5,25
ANSYS Academic Fuel Cell Tools	1,5,25

Core Academic Product Levels



License Borrow

- Available for ANSYS Academic Teaching “Structural & Fluid Dynamics” products only.
- Allows one or more tasks from a multiple task license to be borrowed.
- n-1 tasks may be borrowed from an n task license
- Maximum borrow duration is 1 week.
- Early borrow return is enabled.
- MCAD connections will have the borrow capability.
- The “unlimited” DesignSpace capability does NOT have the borrow capability.
- Borrow is turned OFF by default.
- It is turned on by request, is free & requires an End User certification form & additional language on the license form.



ANSYS

Workbench System Level Coupling

SVS FEM
Your partner in computing

•**For ANSYS Academic products, system coupling is available for the following license combinations:**

- **2 or more tasks of ANSYS Academic Research, Associate, Teaching Intro, Teaching Advanced,**
  - **User must switch to non shared mode under Preference settings.**
  - **2 tasks are consumed for each Mechanical-Fluent system coupling.**
- **1 task of ANSYS Academic Research CFD & 1 task of ANSYS Academic Research**
- **1 task of ANSYS Academic Research CFD & 1 task of ANSYS Academic Research Mechanical**
- **Same combinations for Associate & Teaching products.**

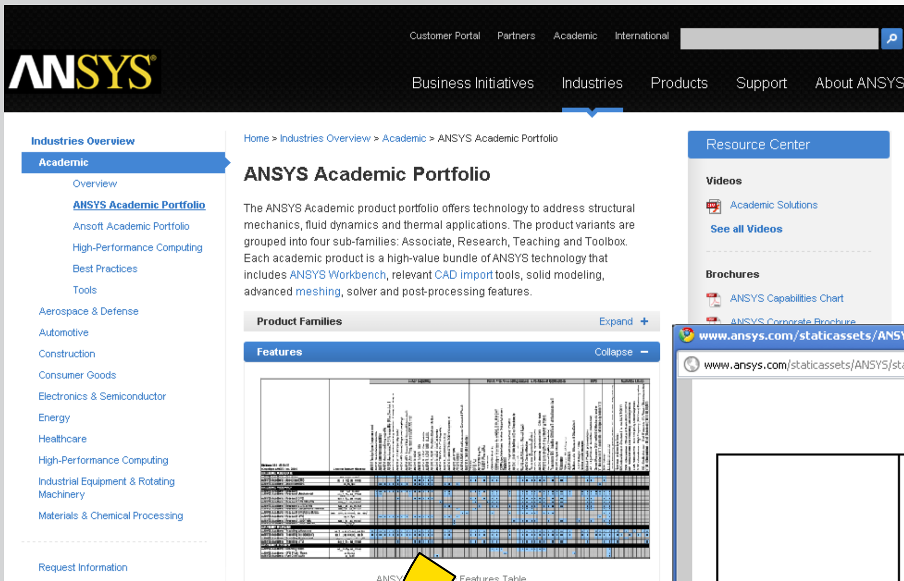
New at 14.0

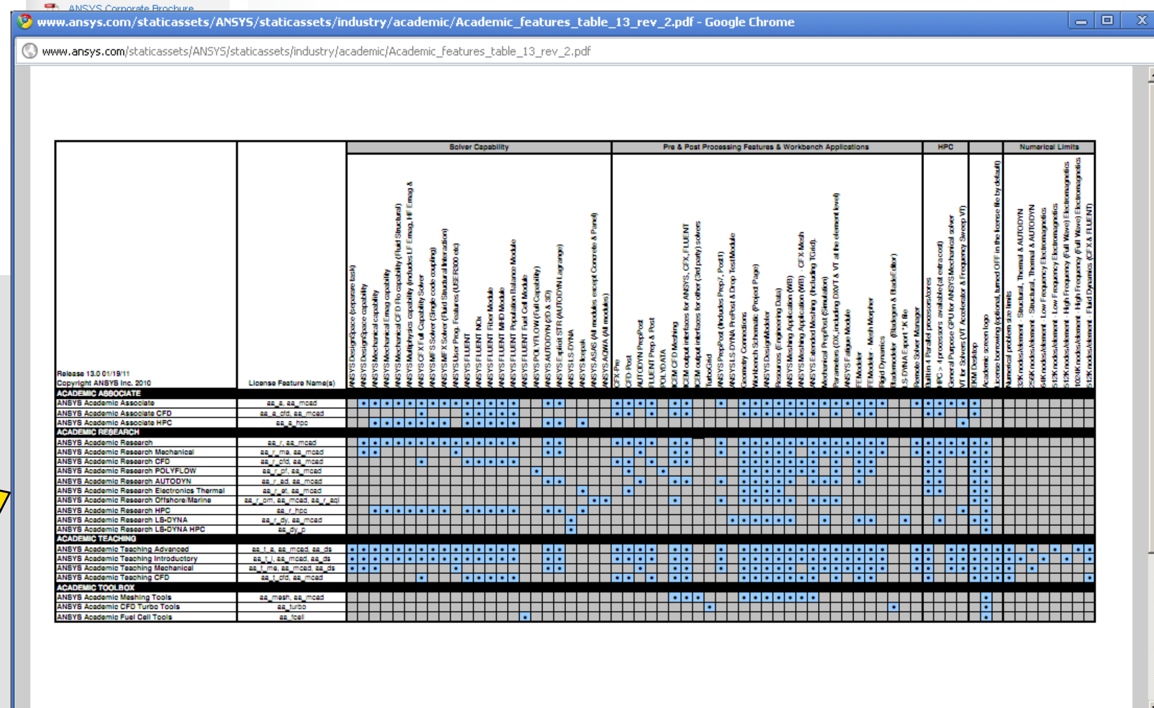
•*Note: System Coupling is not supported by a single task Academic license, at least two tasks must be available to a user as described above!*

14.0 Academic Product Features Table:

SVS FEM
Your partner in computing

- http://www.ansys.com/Industries/Academic/ANSYS+Academic+Portfolio





  November 25, 2013

Student Portal Access Privileges

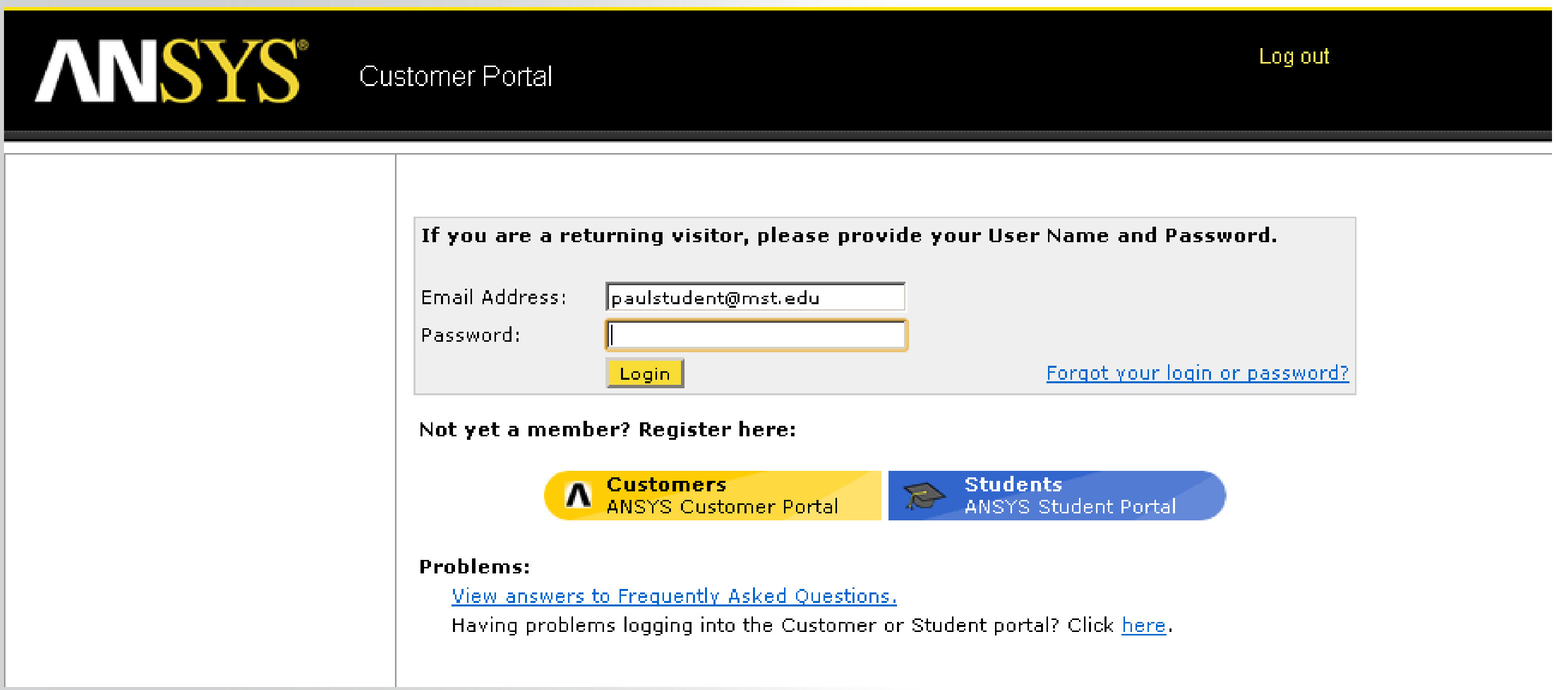
Access Level	View Technotes, FAQ's Knowledge resources	View & Download product document ation	Submit technical support requests	Download all products & service packs	Download ANSYS Academic Student	Access Student Specific pages	Access Fluent USC	Access Ansoft OTS
Direct Customer ASC, Admin, Prof, User	YES	YES	YES	YES	NO	NO	YES	YES
Direct Student	YES	YES	YES*	NO	YES**	YES	NO	YES
Indirect Customer ASC, Admin, Prof, User	YES	YES	NO	YES	NO	NO	YES	YES
Indirect Student	YES	YES	NO	NO	YES**	YES	NO	YES

Student Registration:

- There is a student specific registration page & process:
- During registration, students are required to identify their professor (from a drop down selection) associated with their academic product license, this acts as an important verification mechanism, plus it allows us to tie registered students to specific accounts & professors.
- The drop down selection of professors is populated from our database.
- Each academic account ANSYS Customer Portal user has the option to check “Professor” in their “My Account” settings of the ANSYS Customer Portal.
- Students from a given university will only be able to register if there is at least one associated “Professor” in our database.
- Registered students will have a user type of “Student” rather than “Customer” and their registration will be active for one year.

Student Portal Login Page

- Clear differentiation between the two portals when registering.
- Login page is the same regardless.
- Once logged in the content changes to student specific



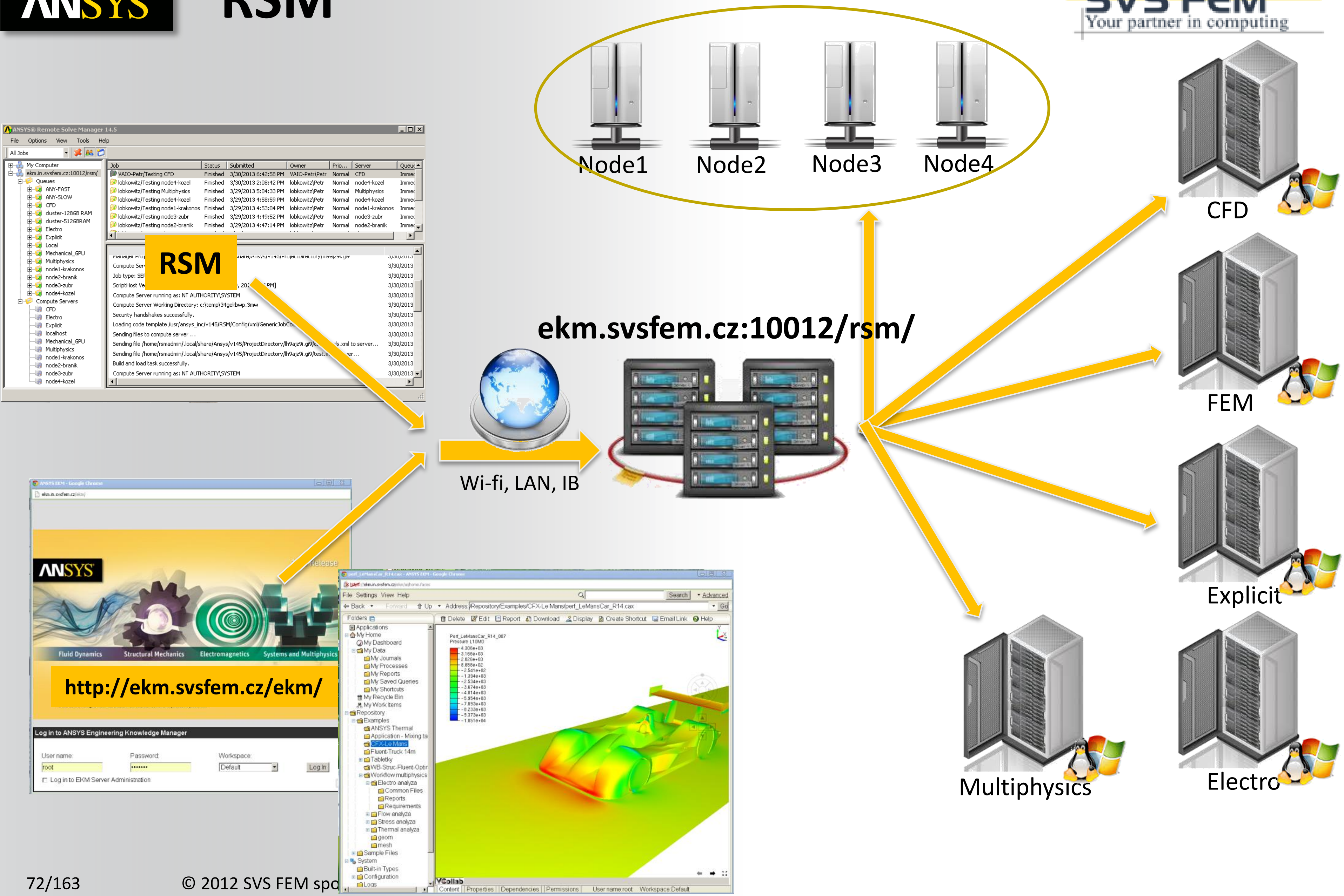
Budoucnost ANSYS HPC

- Aktuální vývoj v ANSYSu
  – HPC technologie pro etrémně rozsáhlé clustery/cloudy
    - Rovnoměrný výkon mezi 100-10000 jader pro FEM i CFD
    - Nové řešiče pro FEM: Multilevel PCG, 2D parallel DSPARSE fronts
    - GP-GPUs pro radiaci, UDFs, DEM a další CFD řešiče
    - Hybridní distribuované/sdílené a vektorové technoogie HPC
    - Škálovatelnost přes všechny komponenty a během celého procesu simulace
    - Síťování, nastavení, řešič, I/O, vizualizace, optimalizace,...
    - Integrace distribuovaného paralelního meshingu spolu s řešičem
    - Paralelizace pro lineární dynamiku včetně superpozičních metod
  – Optimalizace výkonu
    - Dynamické vyvážení zátěže, optimalizované mapování zdrojů, optimalizace kompilátoru
  – Použitelnost
    - Prostředí pro více-složkovou paralelizaci, podpora plánovačů
    - Nástroje pro vyšší toleranci chybovosti HW, dohled a debugging

HPC současnost

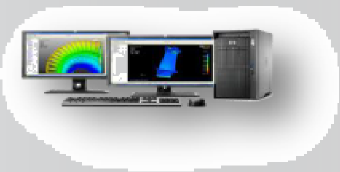
Direct sparse solver – vylepšený algoritmus pro paralelizaci dekompozičních metod (LU) - METIS/ParMETIS
Iterační PCG solver – paralelní odvozování předpodmiňovačů
ANSYS Fluent - Dělení úlohy podle přenosových charakteristik sítě
ANSYS Fluent – Hybridní paralelismus (OpenMP (shared memory)+MPI (distributed))
ANSYS 14 – zkompilován v Intel compileru – využívá nové AVX instrukční sady procesorů Intel/AMD
ANSYS 14 – zavedena paralelizace I/O operací

RSM



Node1
Node2
Node3
Node4
RSM
ekm.svsfem.cz:10012/rsm/
Wi-fi, LAN, IB
http://ekm.svsfem.cz/ekm/
CFD
FEM
Explicit
Multiphysics
Electro

Advantages/Disadvantages



Local computing

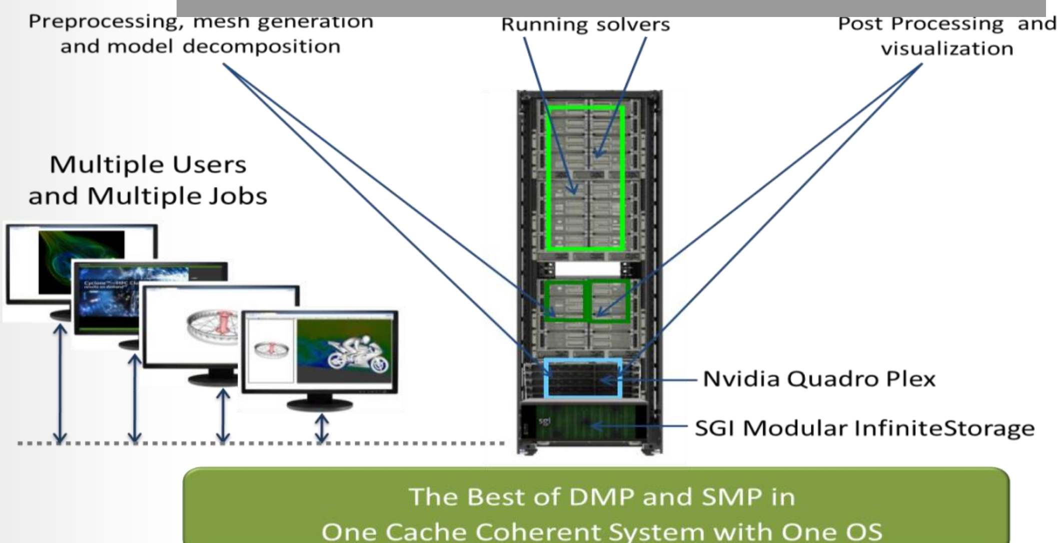
- Pre-processing/ solve/ post-processing on local desktop system
- Files stored locally under individual control
- Inherent capacity limitations; also limits collaboration and data management

Centralized computing with interactive remote access

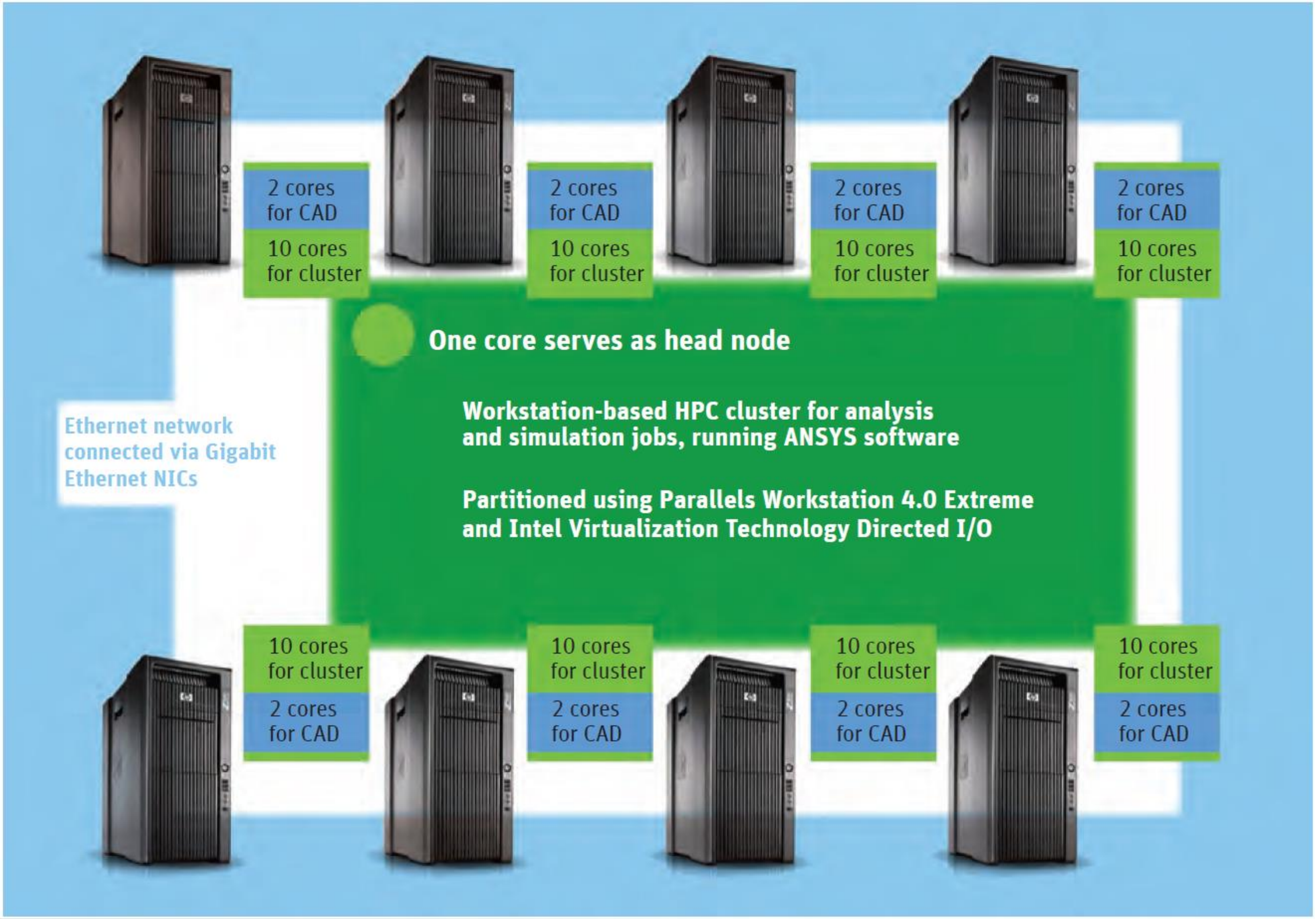
- Solver execution conducted on central remote HPC resource
- Pre-processing/post-processing also conducted remotely, utilizing thin client technology
- Simulation files kept centrally, so bottlenecks related to file transfer minimized
- Limitations of local hardware minimized such as the inability to post-process large files on the local machine
- Emerging remote access and job management solutions enhance collaboration and data management

SVS FEMWorks

SGI UV2 from SVS FEM

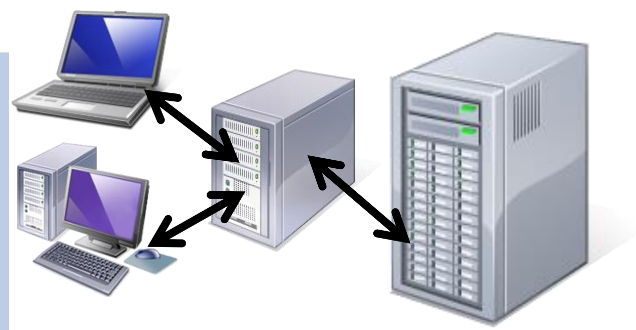


HPC – virtuální cluster



Kenneth Wong ,Taming the HPC Queue, ANSYS ADVANTAGE, Volume V, Issue 3, 2011

ANSYS Remote Solve Manager



Remote Solve Manager (RSM)

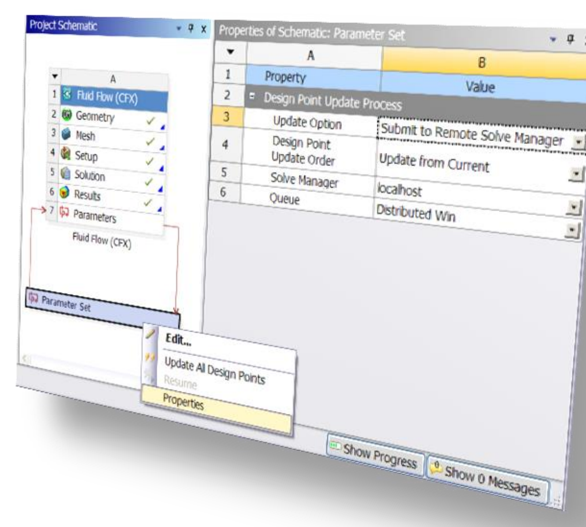
- ►Three-tiered architecture
  - ►Client, solve manager, compute server
- ►Supports third-party schedulers
  - ►LSF, PBS, MSCC

Solution Cell Update

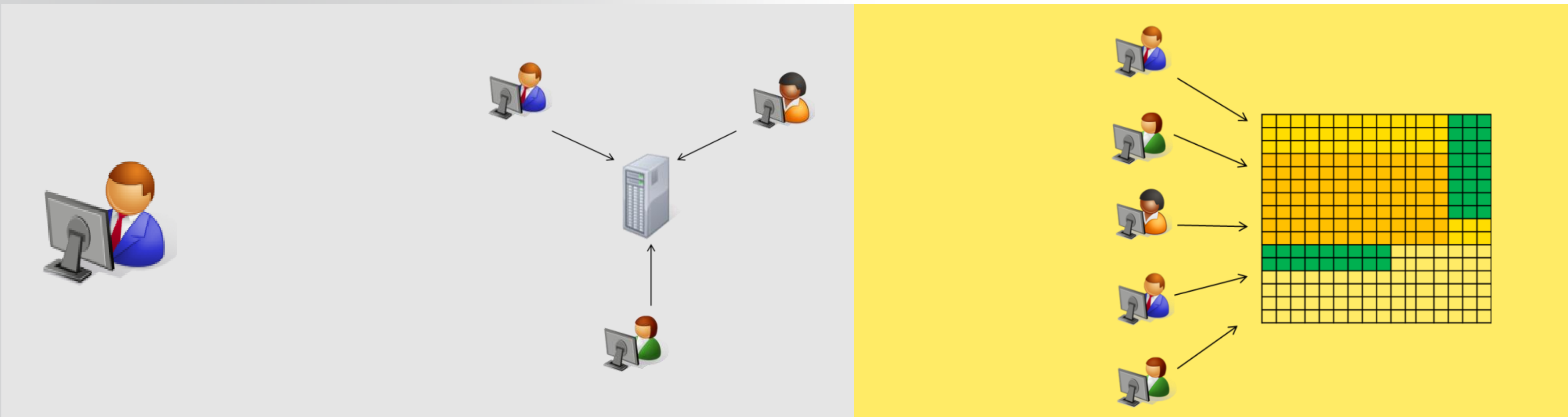
- ►Mechanical, MAPDL , Fluent, CFX and Polyflow

Design Point Update

- ►All design points can be packaged for solution via RSM.



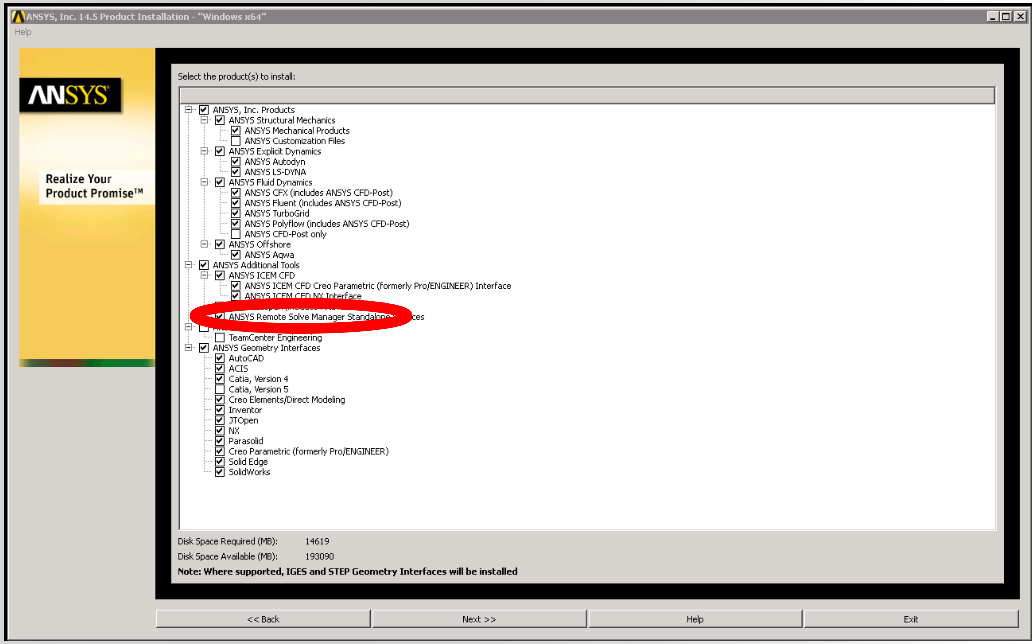
Where does RSM Fit in the workflow?



Desktop | Server | Cluster (with 3rd party scheduler)

RSM as a scheduler	RSM as a transport Mechanism
• Submits to RSM itself. • Unit recognition: Jobs • E.g. a run of a solver such as CFX, FLUENT or Mechanical	• Submits through RSM to a high level scheduler such as LSF, PBS or HPC 2008. • Unit recognition: cores

Instalace a konfigurace RSM

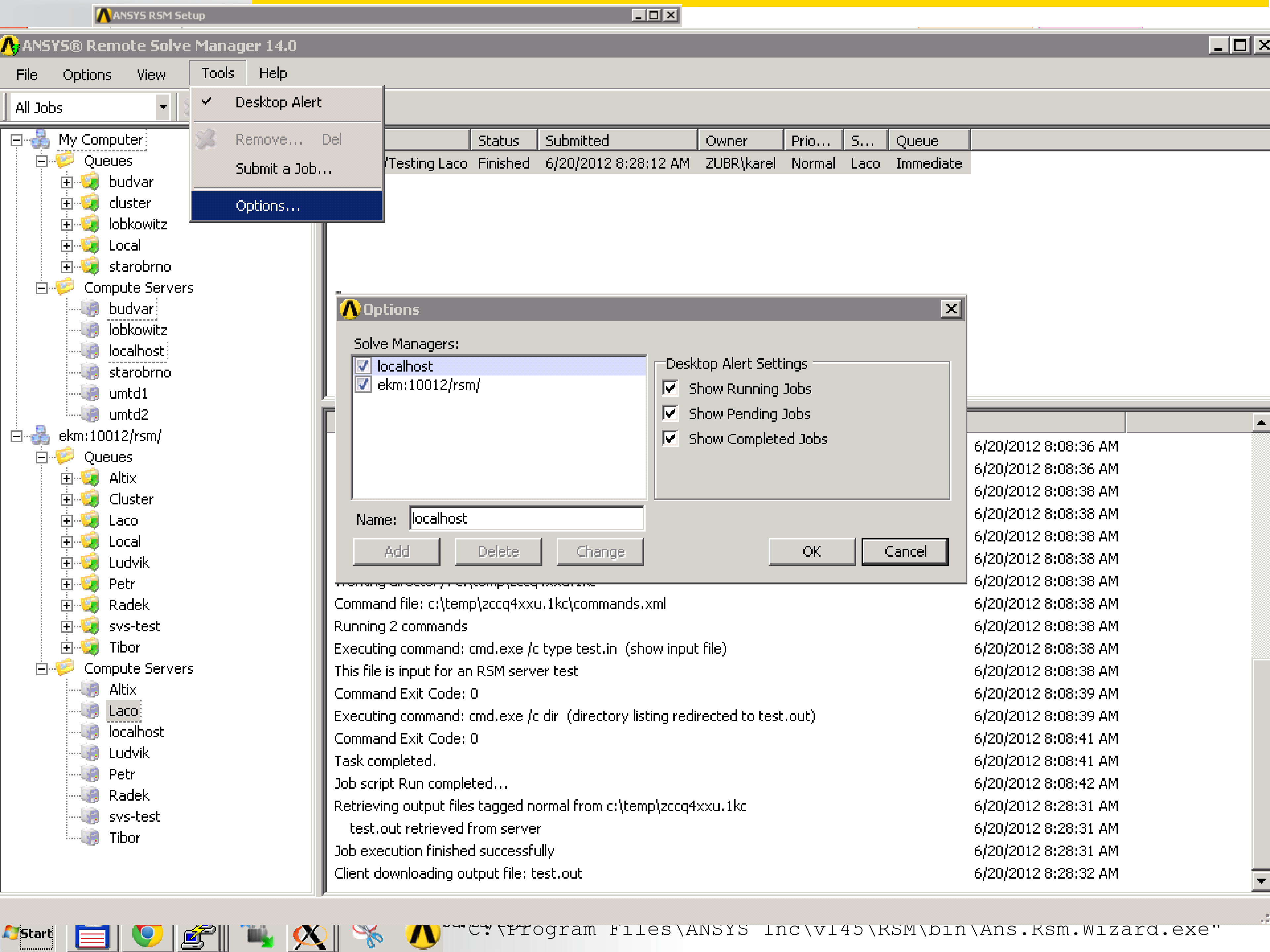


Při instalaci ANSYSu je nutné zvolit
ANSYS Remote Solve Manager Standalone Services

Jako administrátor:

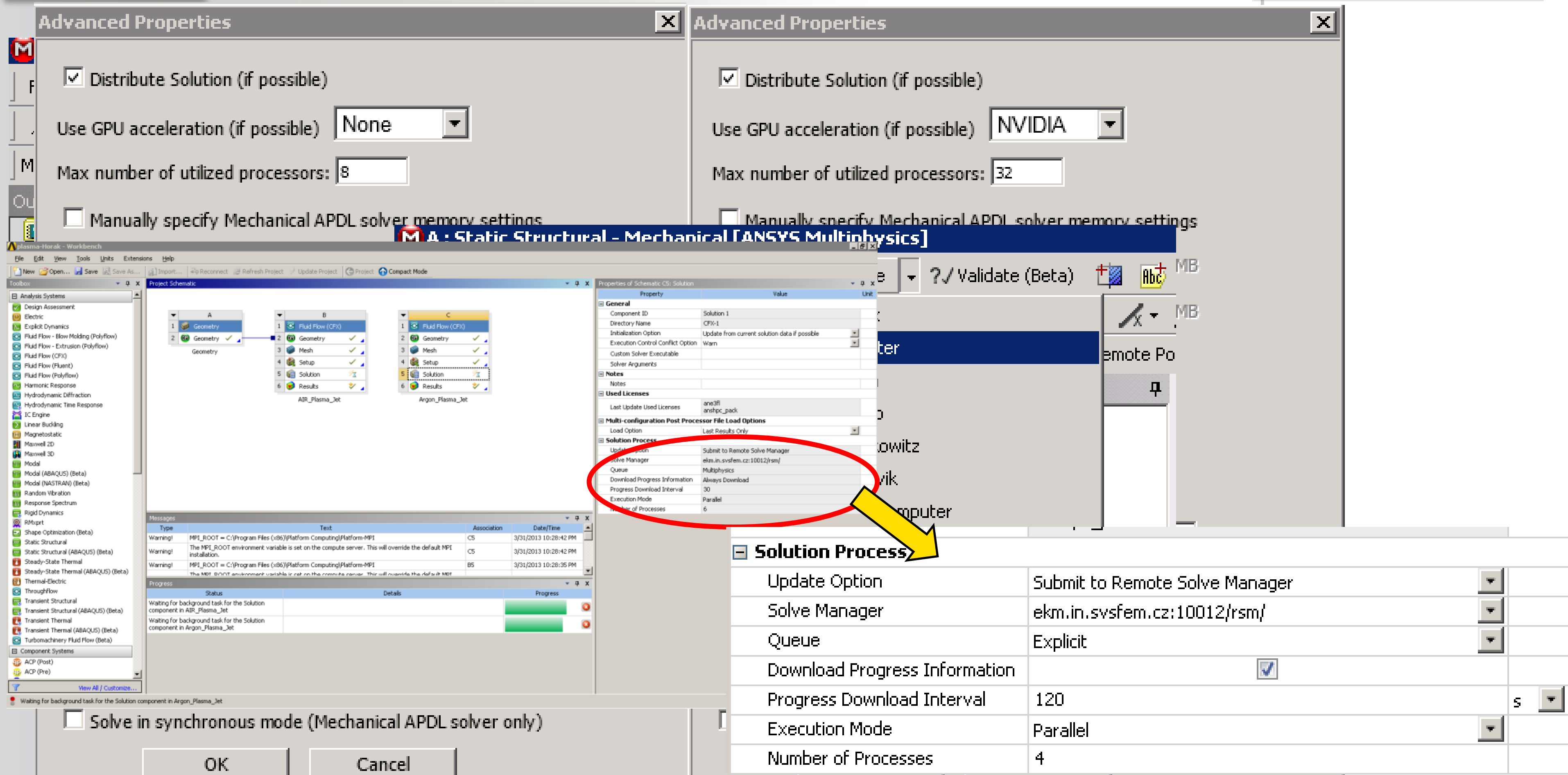
```
"c:\Program Files\ANSYS Inc\v145\RSM\bin\AnsConfigRSM.exe"  -mgr -svr

Nainstalují se služby:
```

ANSYS JobManager service-"c:\Program Files\ANSYS Inc\v145\RSM\bin\Ans.Rsm.JMHost.exe"
Komunikuje na portu 8145/tcp
ANSYS ScriptHost service-"c:\Program Files\ANSYS Inc\v145\RSM\bin\Ans.Rsm.SHHost.exe„
Komunikuje na portu 9145/tcp



ANSYS RSM Setup
ANSYS® Remote Solve Manager 14.0
File
Options
View
Tools
Help
All Jobs
Desktop Alert
Remove... Del
Submit a Job...
Options...
My Computer
Queues
budvar
cluster
lobkowitz
Local
starobrno
Compute Servers
budvar
lobkowitz
localhost
starobrno
umtd1
umtd2
ekm:10012/rsm/
Queues
Altix
Cluster
Laco
Local
Ludvik
Petr
Radek
svs-test
Tibor
Compute Servers
Altix
Laco
localhost
Ludvik
Petr
Radek
svs-test
Tibor
Status
Submitted
Owner
Prio...
S...
Queue
/Testing Laco Finished 6/20/2012 8:28:12 AM ZUBR\karel Normal Laco Immediate
Options
Solve Managers:
localhost
ekm:10012/rsm/
Desktop Alert Settings
Show Running Jobs
Show Pending Jobs
Show Completed Jobs
Name: localhost
Add
Delete
Change
OK
Cancel
Command file: c:\temp\zccq4xxu.1kc\commands.xml
Running 2 commands
Executing command: cmd.exe /c type test.in (show input file)
This file is input for an RSM server test
Command Exit Code: 0
Executing command: cmd.exe /c dir (directory listing redirected to test.out)
Command Exit Code: 0
Task completed.
Job script Run completed...
Retrieving output files tagged normal from c:\temp\zccq4xxu.1kc
test.out retrieved from server
Job execution finished successfully
Client downloading output file: test.out
6/20/2012 8:08:36 AM
6/20/2012 8:08:36 AM
6/20/2012 8:08:38 AM
6/20/2012 8:08:38 AM
6/20/2012 8:08:38 AM
6/20/2012 8:08:38 AM
6/20/2012 8:08:38 AM
6/20/2012 8:08:38 AM
6/20/2012 8:08:38 AM
6/20/2012 8:08:38 AM
6/20/2012 8:08:38 AM
6/20/2012 8:08:39 AM
6/20/2012 8:08:39 AM
6/20/2012 8:08:41 AM
6/20/2012 8:08:41 AM
6/20/2012 8:08:42 AM
6/20/2012 8:28:31 AM
6/20/2012 8:28:31 AM
6/20/2012 8:28:31 AM
6/20/2012 8:28:32 AM
Start
"C:\Program Files\ANSYS Inc\v145\RSM\bin\Ans.Rsm.Wizard.exe"

Konfigurace HPC pro WB

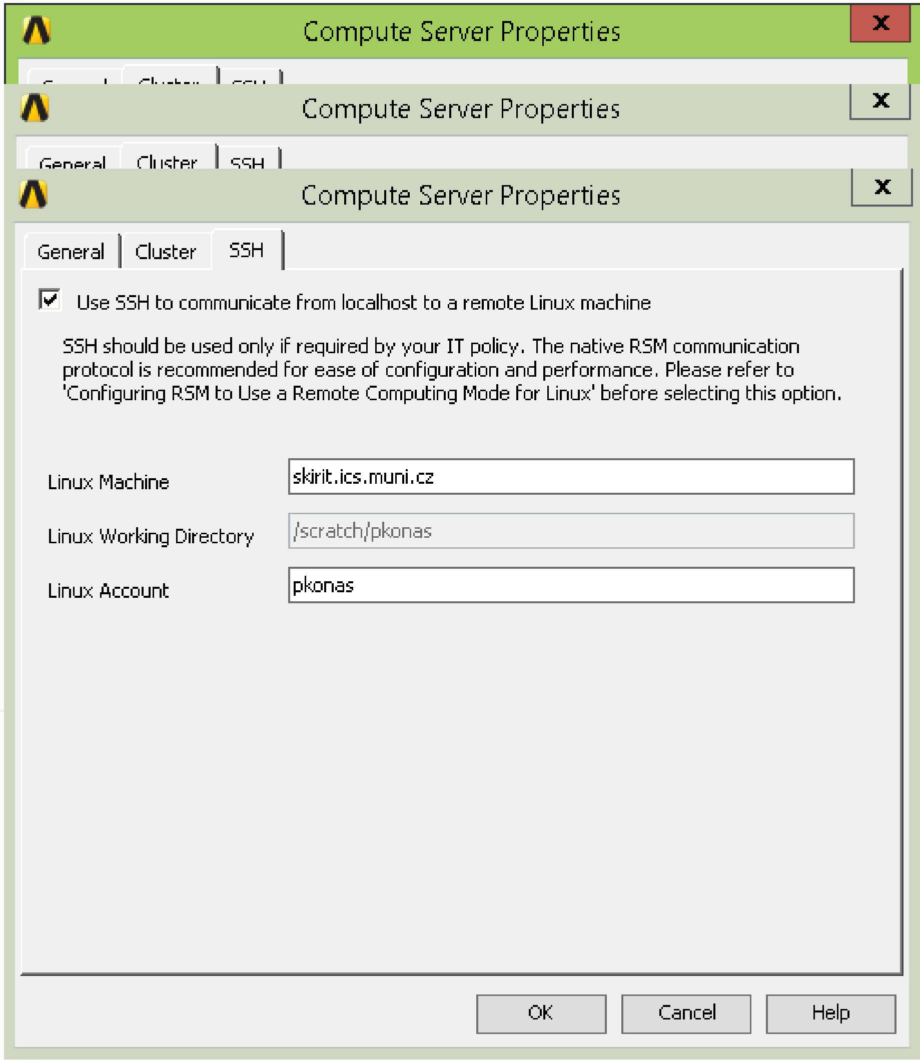


Lokální či vzdálený výpočet

Distribuovaný výpočet

Integrace s PBS/LSF

```
#!/bin/bash
#PBS -l nodes=2:ppn=12
#
#For Stoney, you must use ppn=8
#e.g. for a 16-core job:
##PBS -l nodes=2:ppn=8
#
#PBS -l walltime=25:00:00
#PBS -N MyCFXJobName
#PBS -A MyProjectName

#Load the ansys module - CFX is then accessible
module load ansys

cd $PBS_O_WORKDIR

#Create a list of available nodes for the CFX executable:
nodes=`cat $PBS_NODEFILE`
nodes=`echo $nodes | sed -e 's/ /,/g'`

#Partition + Solve:
cfx5solve -def test.def -par-dist $nodes -start-method 'HP MPI
```



Checking license availabality

```
#PBS -W x=GRES:aa_r_hpc+M
```

where M is the number of license tokens required for the job.

This job can be submitted using the command

```
qsub scriptname.pbs
```

Konfigurace HPC – low level

Základní podmínkou je funkční MPI (`telnet localhost 8636` **pro Platform-MPI)**
Pro Workbench je podmínkou funkční RSM (RSM se musí nastavit vždy po každé nové verzi)

ANSYS krom centralizovaného řešení RSM nabízí i samostatnou podporu HPC jednotlivých komponent nazávisle na RSM

ANSYS Mechanical APDL: `"c:\Program Files\ANSYS Inc\v145\ansys\apdl\hosts145.ans"`
CFX : `"c:\Program Files\ANSYS Inc\v145\CFX\config\hostinfo.ccl"`
Fluent: `hosts file, Microsoft Job Scheduler`
LS-DYNA: `hosts file`

Konfigurace HPC pro ANSYS APDL

Konfigurace hosts145.ans

lobkowitz winx64 0 4 0 0 c:\temp MPI 1 1
budvar winx64 0 4 0 0 c:\temp MPI 1 1
starobrno winx64 0 4 0 0 c:\temp MPI 1 1
prazdroj winx64 0 4 0 0 c:\temp MPI 1 1
zasedacka winx64 0 2 0 0 c:\temp MPI 1 1

ANSYS Classic

"c:\Program Files\ANSYS Inc\v145\ansys\apdl\hosts145.ans"

Konfigurace HPC pro CFD (MPI)

SVS FEM
Your partner in computing

Struktura hostinfo.ccl
Struktura hosts z fluentu, ls-dyna

Lobkowitz
Budvar
Starobrno
Prazdroj
zasedacka

Fluent, LS-DYNA

../hosts

CFX

"c:\Program Files\ANSYS Inc\v145\CFX\config\hostinfo.ccl"

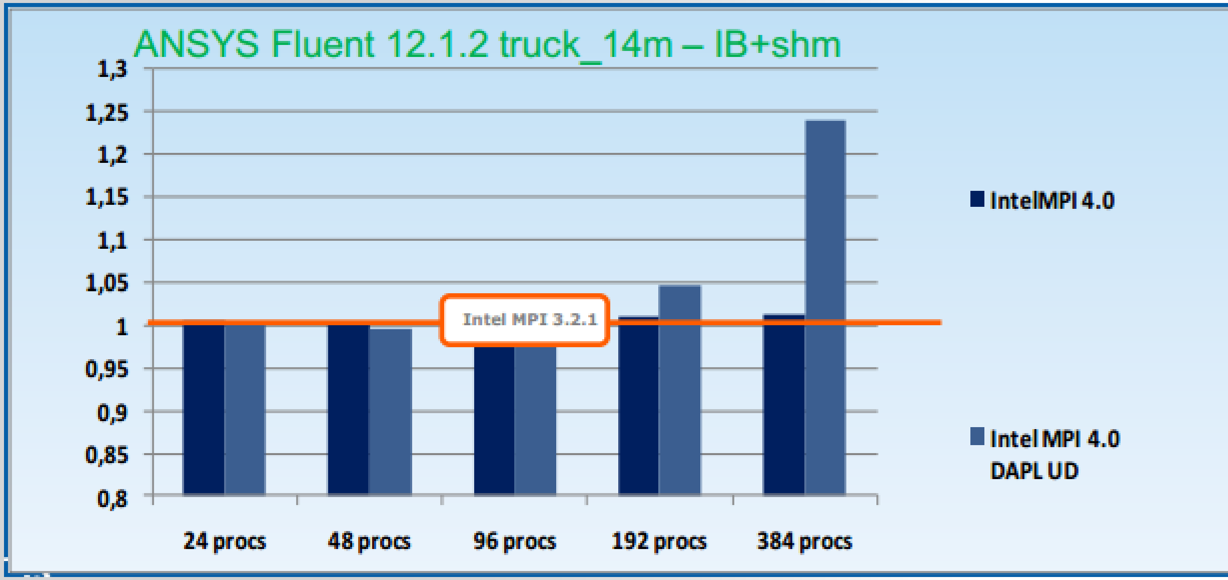
```
SIMULATION CONTROL:
  EXECUTION CONTROL:
    PARALLEL HOST LIBRARY:
      HOST DEFINITION: starobrno
        Installation Root = C:\Program Files\ANSYS Inc\v%v\CFX
        Host Architecture String = winnt-amd64
                        Relative Speed = 11.94
                        Number of Processors =8
      END # HOST DEFINITION starobrno
      HOST DEFINITION: lobkowitz
        Installation Root = C:\Program Files\ANSYS Inc\v%v\CFX
        Host Architecture String = winnt-amd64
                        Relative Speed = 11.94
                        Number of Processors =8
      END # HOST DEFINITION lobkowitz
      HOST DEFINITION: budvar
        Installation Root = C:\Program Files\ANSYS Inc\v%v\CFX
        Host Architecture String = winnt-amd64
                        Relative Speed = 11.94
                        Number of Processors =8
      END # HOST DEFINITION budvar
      HOST DEFINITION: bernard2
        Installation Root = C:\Program Files\ANSYS Inc\v%v\CFX
        Host Architecture String = winnt-amd64
                        Relative Speed = 11.94
                        Number of Processors =8
      END # HOST DEFINITION bernard2
      HOST DEFINITION: prazdroj
        Installation Root = C:\Program Files\ANSYS Inc\v%v\CFX
        Host Architecture String = winnt-amd64
                        Relative Speed = 11.94
                        Number of Processors =8
      END # HOST DEFINITION prazdroj
END # PARALLEL HOST LIBRARY
  END # EXECUTION CONTROL
END # SIMULATION CONTROL
```

Optimální nastavení úlohy pro HPC (MPI) - tunning

Interconnect prot.: Ib, dapl, tcp, shm:dapl, ofa, tmi, ofed

- **HP-MPI**
- Message latency and bandwidth (MPI_Pack, MPI_Unpack, MPI_ANY_SOURCE, MPI_Recv_init, MPI_startall, NUMA,…)
- Multiple network interfaces (MPI_TOPOLOGY, R-server, K-server,…)
- Processor subscription (optimalizace komunikace s plánovači RSM, PBS, LSF,...)
- MPI routine selection (multilevel parallelism, process placement,...)
  – **http://www.ncsa.illinois.edu/UserInfo/Resources/Hardware/CommonDoc/HP/MPI/3_understand.html**
- **Intel MPI**
- MPI profile (I_MPI_STATS, I_MPI_STATS_SCOPE)
- Interconnect (I_MPI_FABRICS, I_MPI_ADJUST_REDUCE, I_MPI_DAPL_SCALABLE_PROGRESS,…)
- Layout (I_MPI_PERHOST, I_MPI_PIN_PROCESSOR_LIST,…)
- MPI/OpenMP (OMP_NUM_THREADS)
- Connection mode (I_MPI_DYNAMIC_CONNECTION, I_MPI_WAIT_MODE, I_MPI_SHM_BYPASS,…)
  – **http://www.rz.rwth-aachen.de/global/show_document.asp?id=aaaaaaaaacfigd**

Intel MPI

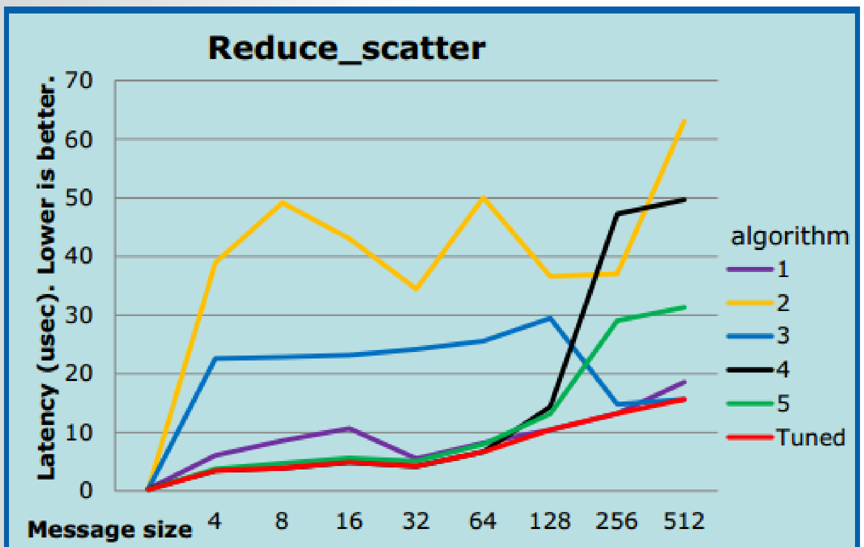


Benchmark overview

- Intel® MPI Library 4.0 options
  - -genv I_MPI_ADJUST_REDUCE 2
  - -genv I_MPI_ADJUST_BCAST 0
  - -genv I_MPI_DAPL_SCALABLE_PROGRESS 1
- Fluent benchmark
  - Truck_14m

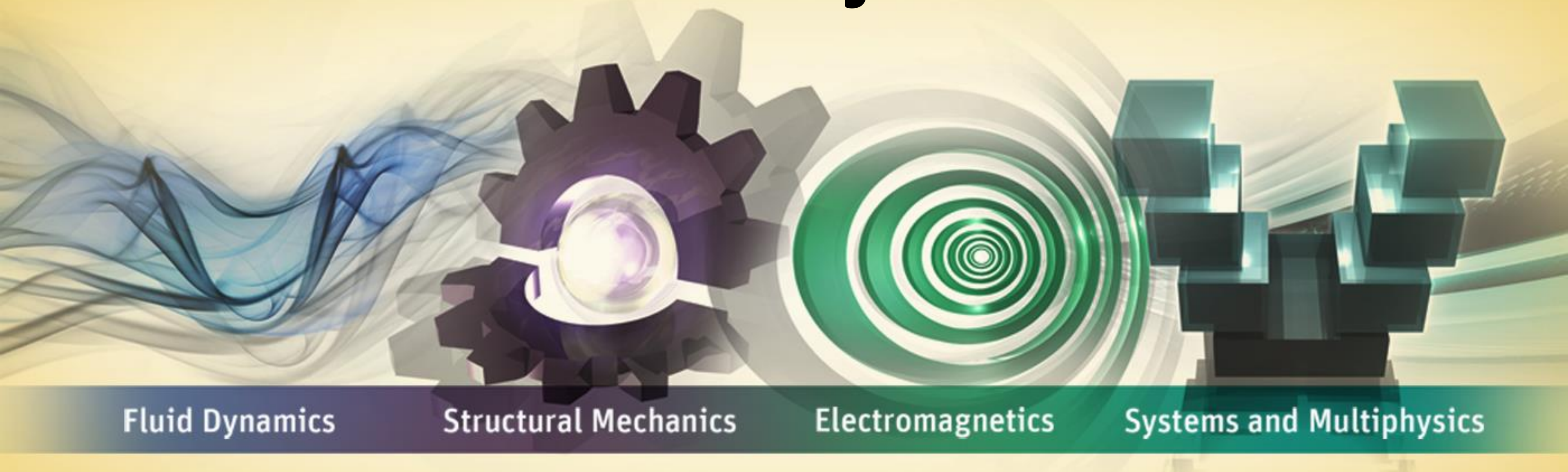
Hardware Configuration:

- Interconnect: InfiniBand, ConnectX adapters; QDR
- CPU: 2.93GHz B0-step Westmere Dual processor (6 cores per processor)
- RAM: 24Gb per system, 1333MHz (0.8ns) DDR3



Realize Your Product Promise™

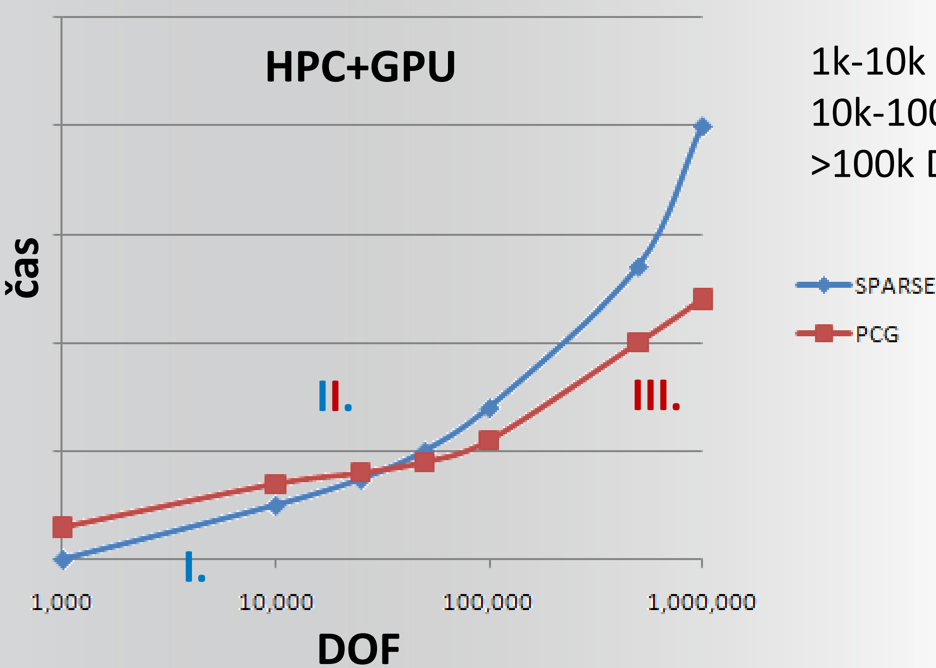
How to select HW resources for ANSYS HPC jobs



SVS FEM
Your partner in computing

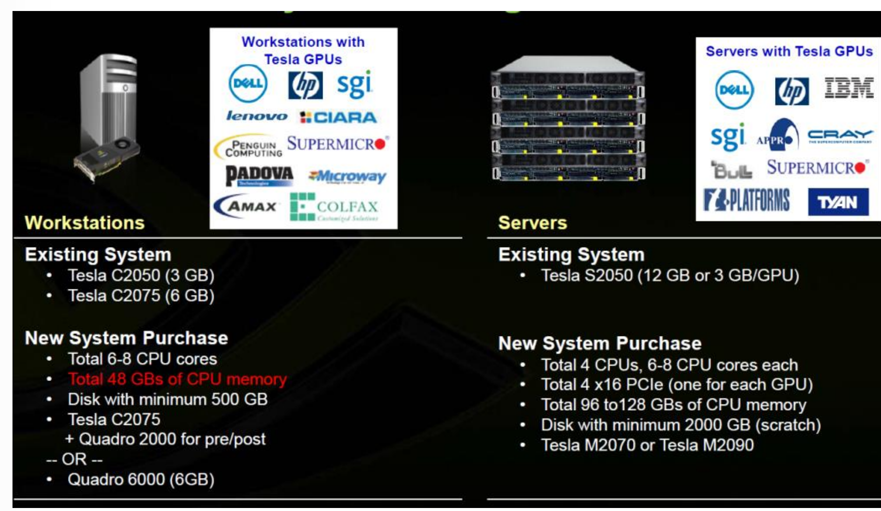
Velikost úloh pro efektivní výpočet

- **Direct sparse: závisí na nejvetší velikosti vstupní fronty: GPU je efektivní pro úlohy od ~1M DOF do ~8M DOF pro 6GB Tesla C2075 or Quadro 6000**
- **Iterační solvery: závisí na velikosti paměti GPU: GPU je efektivní pro úlohy od ~1M DOF do ~5M DOF pro 6GB Tesla C2075 or Quadro 6000**



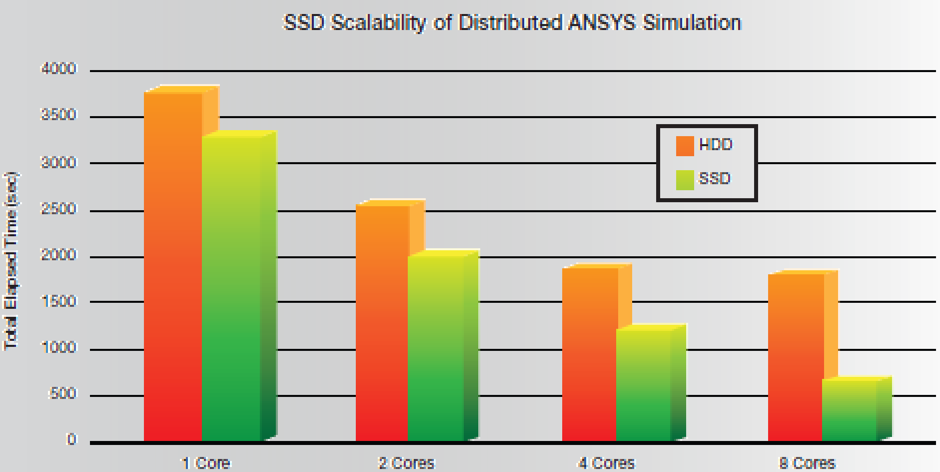
1k-10k DOF rychlejší Sparse
10k-100k DOF Sparse i PCG podobný výkon
>100k DOF rychlejší PCG (méně IO operací)

Pierre Louat , HPC Technology for Structural Simulations, 2012 Sales Conference Theme and Team Building, France 2012

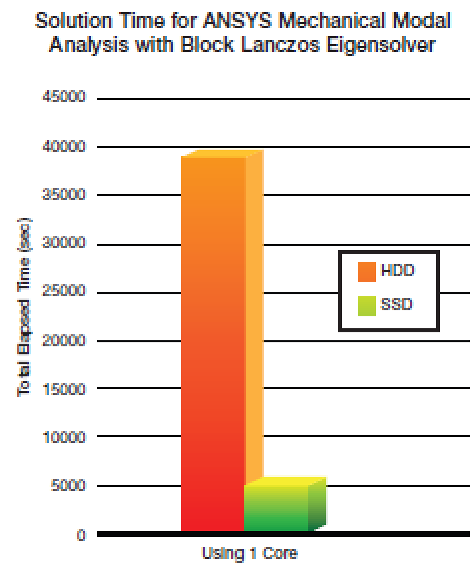


NVIDIA Inc. http://www.nvidia.com

HPC+GPU+SSD



Considerable I/O was performed in an analysis requiring about 30 GB of disk space to run the ANSYS distributed sparse solver on a workstation containing only 24 GB of RAM. The reduced seek times for the SSD significantly improved I/O performance, thus helping to shorten solution time as more cores are involved.



For this study, approximately 1 million degrees of freedom (DOF) were analyzed for 200 frequencies. Elapsed times are compared for simulations on a workstation having two file systems: one with a single SCSI 10k rpm hard disk drive, another with four Intel® X25-E 64 GB SATA SSDs.

Pierre Louat , HPC Technology for Structural Simulations, 2012 Sales Conference Theme and Team Building, France 2012

Kepler GPU Architecture for ANSYS

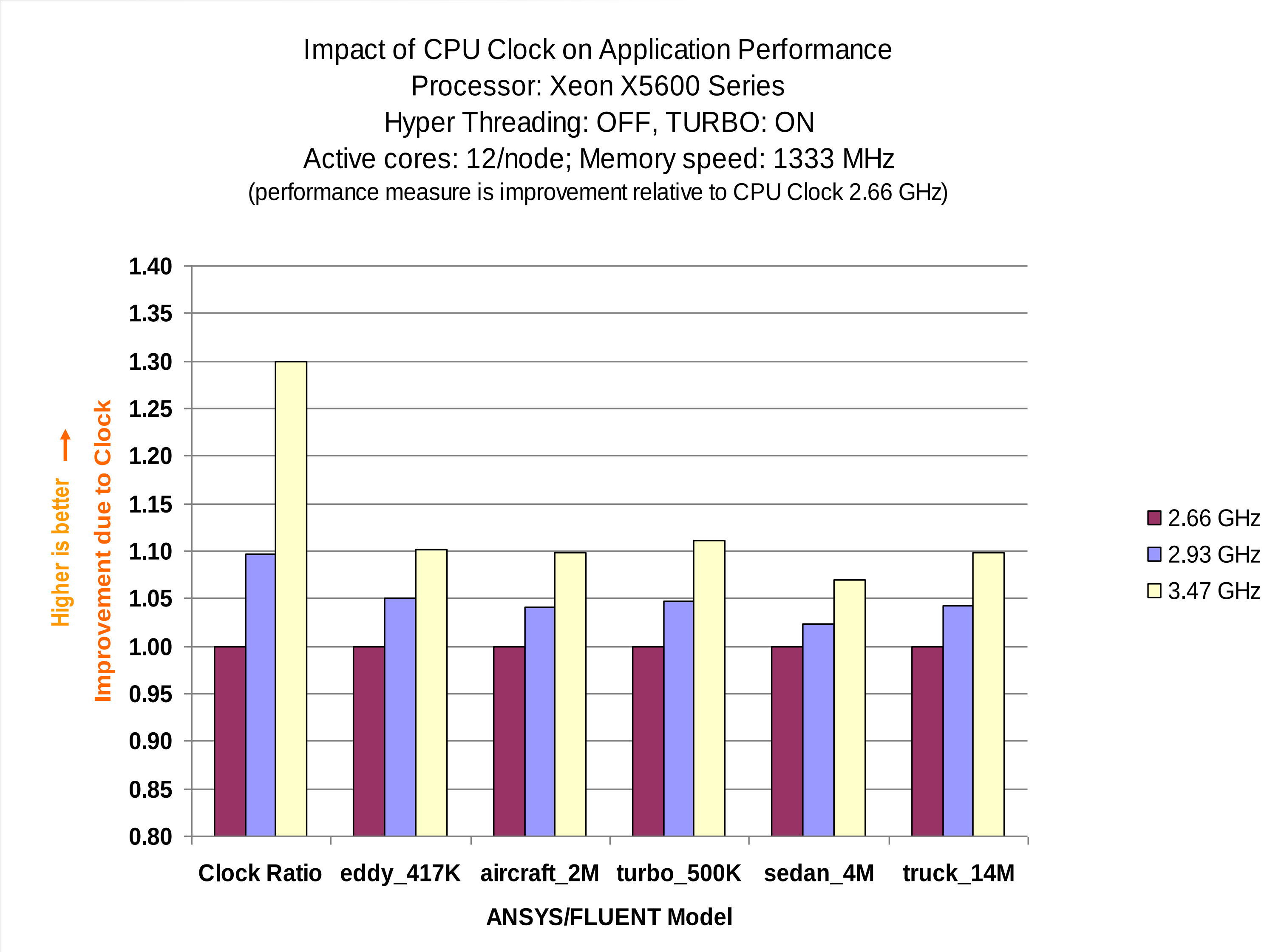
Features	Tesla K20	Tesla C2075
GPU Architecture	Kepler 	Fermi 
DGEMM performance	> 1000 Gigaflops	370 Gigaflops
Memory bandwidth	> 200 GBytes/sec	150 GBytes/sec
Memory size (GDDR5)	6 GigaBytes	6 GigaBytes
CUDA cores	> 1000	448
Available in HP Z8xx MAXIMUS Workstations	Late 2012	Today

ANSYS

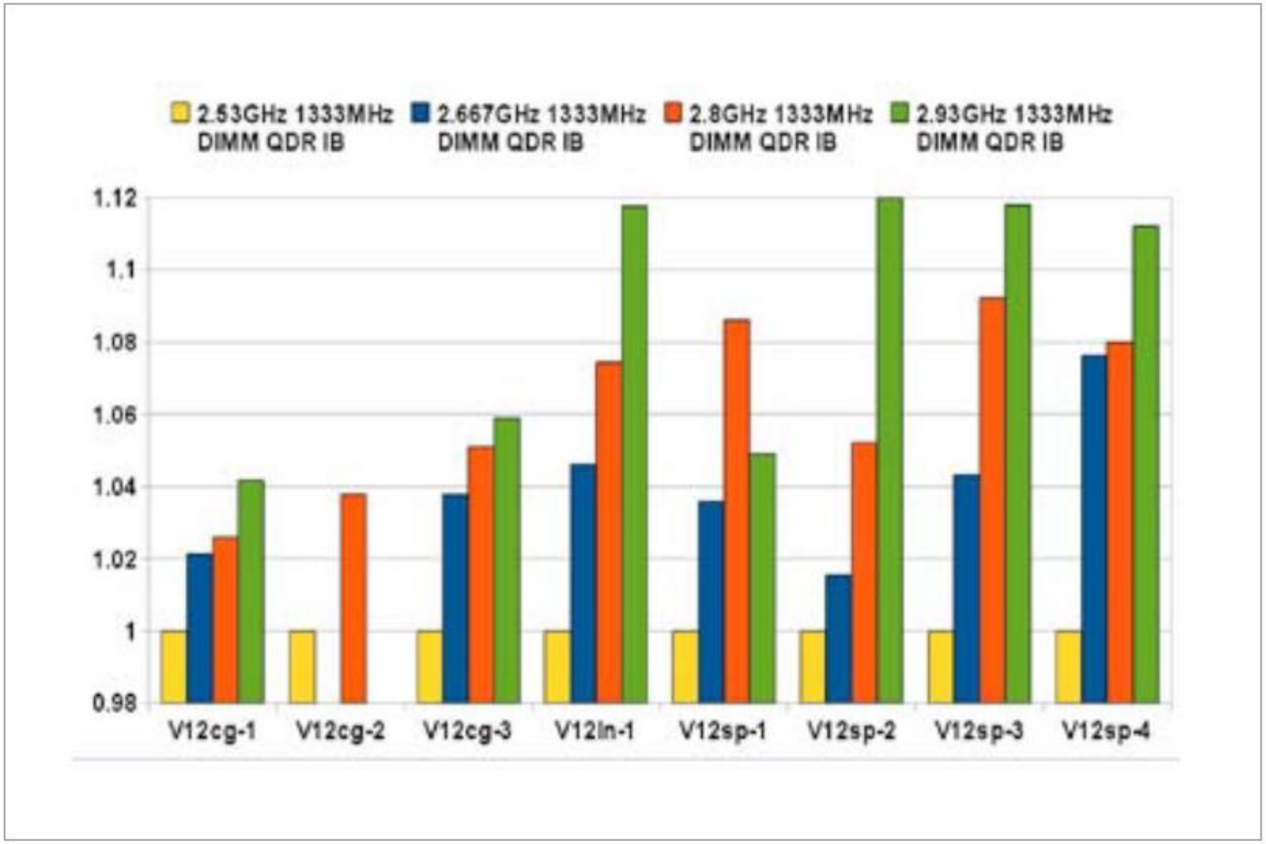
GPU Acceleration Enhancements

- Targeted hardware
  - NVIDIA (-acc nvidia)
    - Tesla 20-series cards are supported (Fermi)
    - Quadro 6000 cards are supported (Fermi)
    - Quadro K5000 cards are supported (Kepler)
    - Quadro K6000 cards should be supported (Kepler)
      - Not formally documented or tested, but expected to work
    - Next-gen Tesla series cards should be supported (Kepler)
      - Not formally documented or tested, but expected to work
      - Tesla K10 cards good for the PCG/JCG iterative solvers only
      - Tesla K20 cards good for all equation solvers

Understanding the effect of clock speed



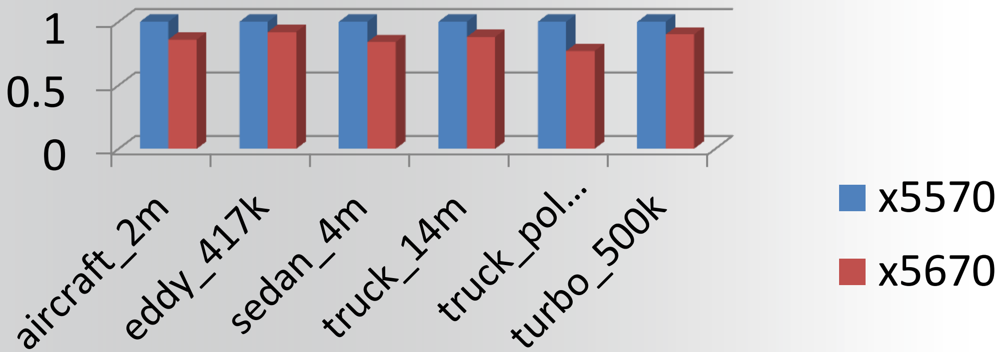
Understanding the effect of clock speed



- **Effect of increased core operating frequencies on the DMP benchmarks running on 8 cores**
- **Influence is highest for sparse solver benchmarks**

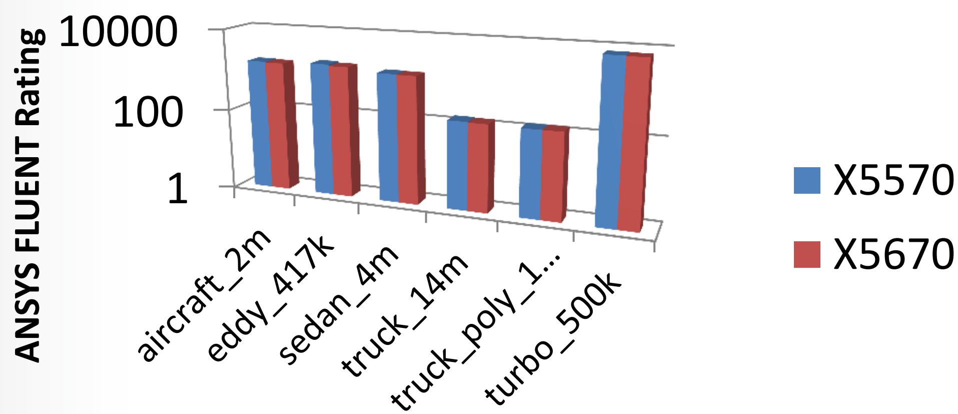
Understanding the effect of memory bandwidth

X5570 quad-core processors have higher performance per core than X5670 six-core processors since X5670 share more resources



Relative performance for 24-way runs: three X5570 nodes, two X5670 nodes

16 of 16 cores used in 2 X5570 nodes
16 of 24 cores used in 2 X5670 nodes



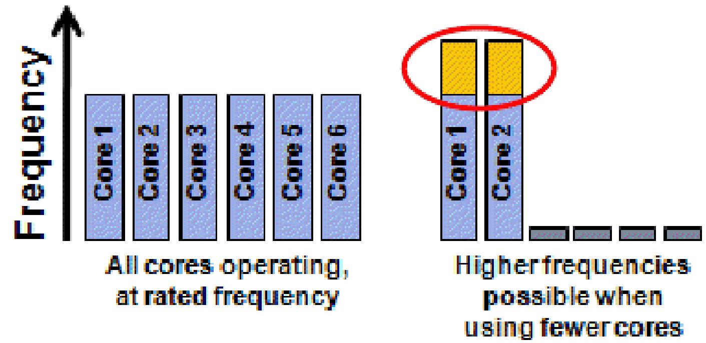
  November 25, 2013

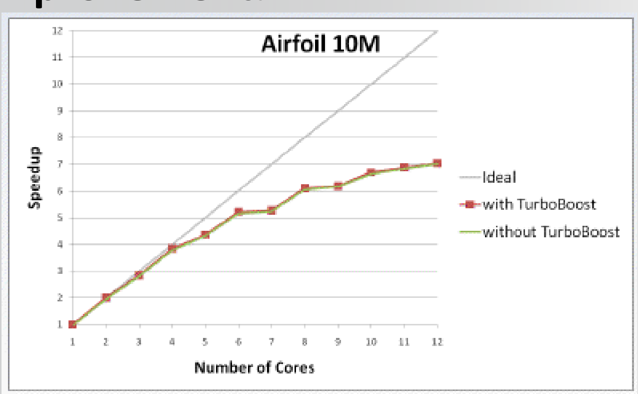
Turbo Boost (Intel) / Turbo Core (AMD)

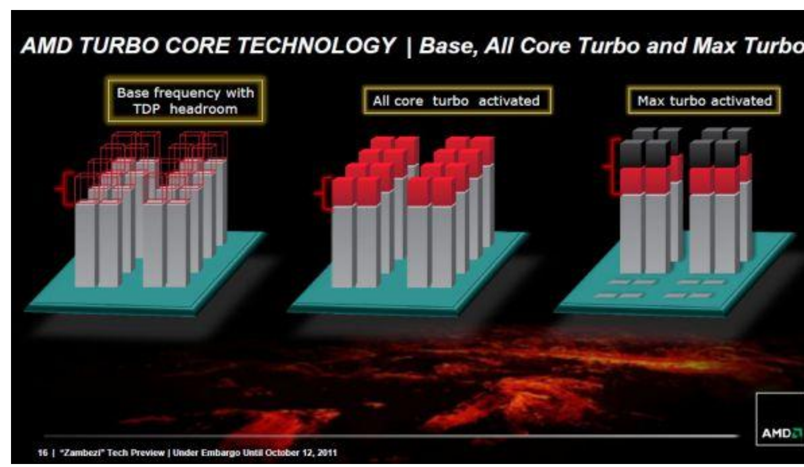
ANSYS

SVS FEM
Your partner in computing

- **Turbo Boost (Intel)/ Turbo Core(AMD) is a form of over-clocking that allows you to give more GHz to individual processors when others are idle.**
- **With the Intel's have seen variable performance with this ranging between 0-8% improvement depending on the numbers of cores in use.**
- **The graph below for CFX on a Intel X5550. This only sees a maximum of 2.5% improvement.**







  November 25, 2013

ANSYS

Turbo Boost (Intel)

SVS FEM
Your partner in computing

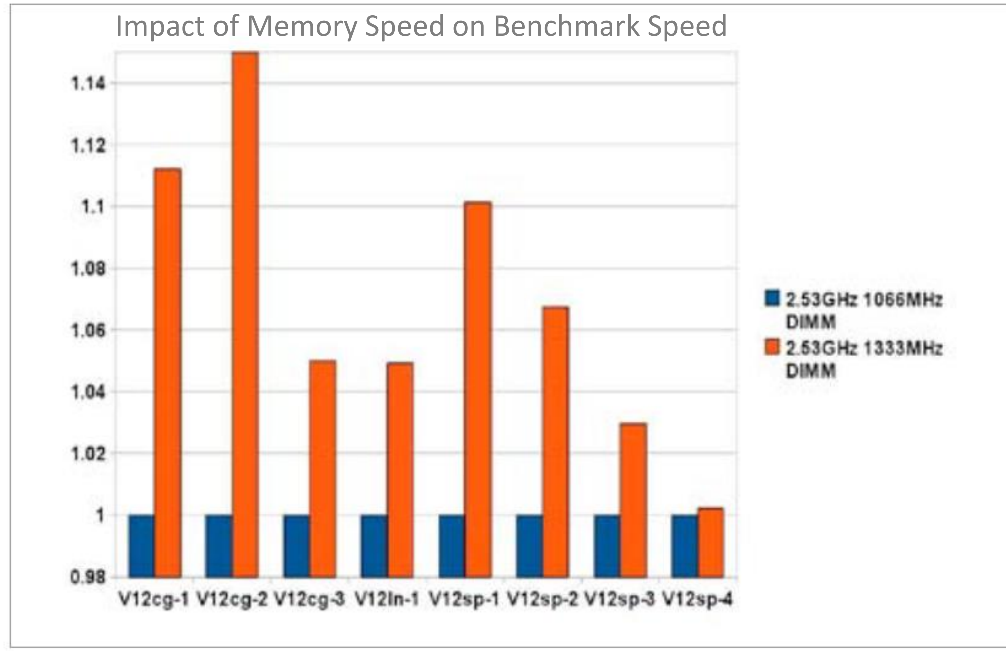
Solver	Number of Cores	Mode	Elapsed Time - Turbo Off	Elapsed Time - Turbo On	Speedup due to Turbo On
Sparse	1	SMP	299	213	1.40
	2		166	124	1.34
	4		121	103	1.17
	1	DMP	296	210	1.41
	2		179	137	1.31
	4		139	117	1.19
PCG	1	SMP	299	231	1.29
	2		207	169	1.22
	4		186	155	1.20
	1	DMP	301	232	1.30
	2		221	186	1.19
	4		180	149	1.21

- **DELL M6600 I7 2820 @ 2.29 GHz**
- **Sparse model : 0.34 M DOF**
- **PCG model: 1.56 M DOF**

Recommendation: Activate Turbo Boost

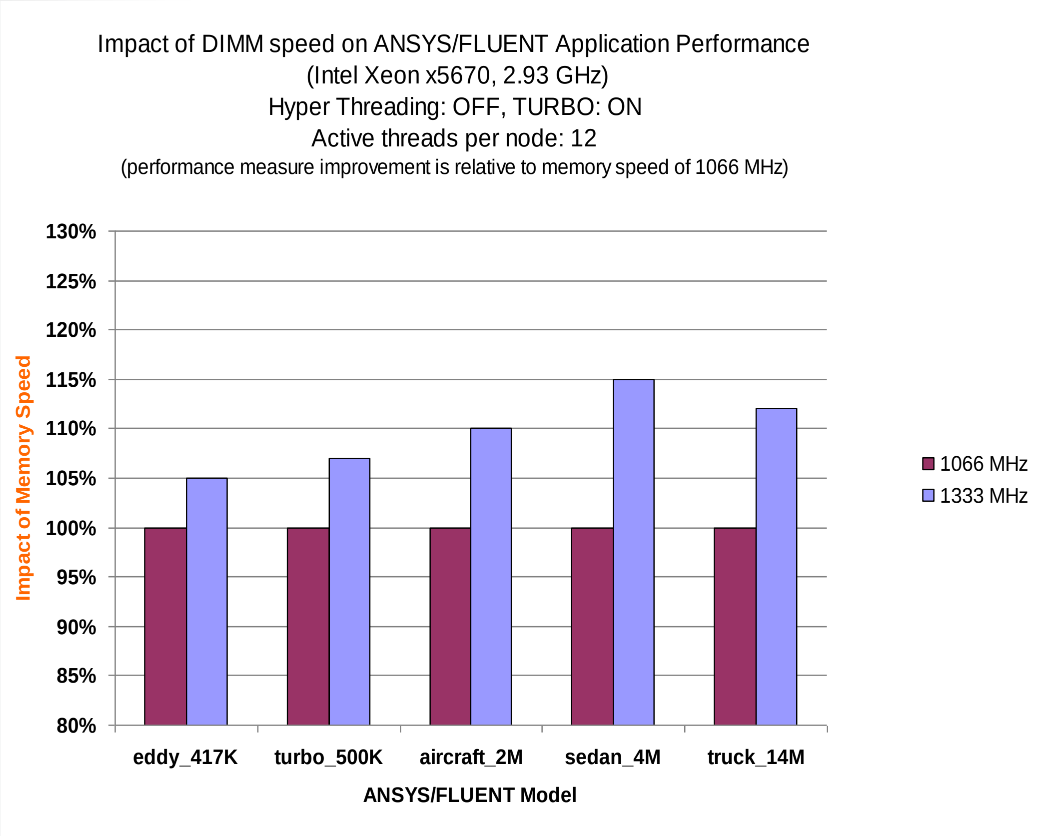
Understanding the effect of memory speed

- We can see here the effect of memory speed.
- This has implications on how you build your hardware.
- Some processors types have slower memory speeds by default.
- On other processors non-optimally filling the memory channels can slow the memory speed.



Understanding the effect of memory speed

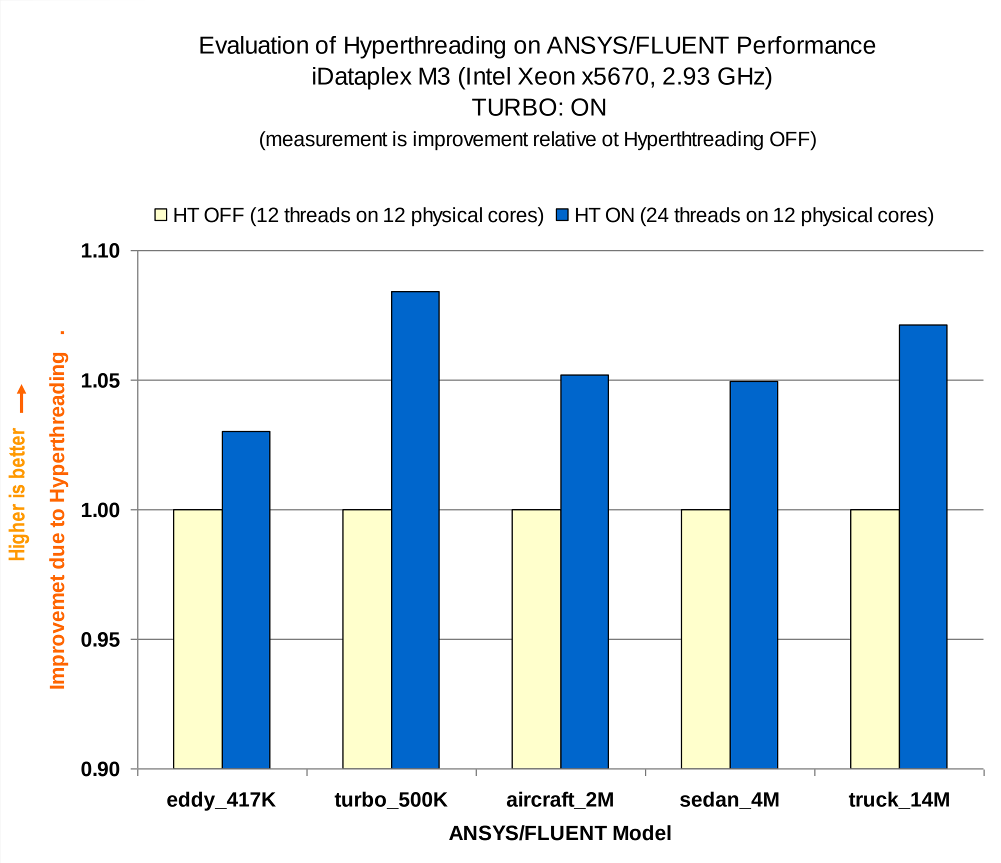
- We can see here the effect of memory speed.
- This has implications on how you build your hardware.
- Some processors types have slower memory speeds by default.
- On other processors non-optimally filling the memory channels can slow the memory speed.



November 25, 2013

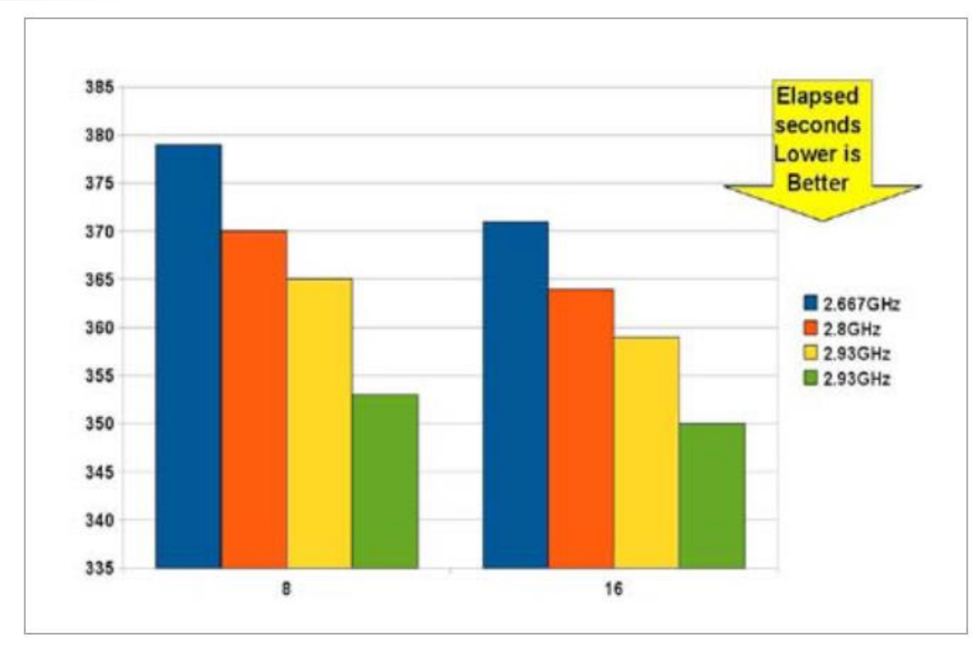
Hyper-threading – ANSYS Fluent

- **Hyper-Threading Technology makes a single physical processor appear as two logical processors.**
- **This is not the same as physically having two logical processors and does not give double the speedup.**
- **In our tests we've seen as high as a 20% increase in performance although you can see the actual performance can be quite variable from the graph opposite.**
- **It is worth noting that this has licensing implications as you would need to oversubscribe the physical cores and hence would need double the HPC Licenses.**



Hyper-threading – ANSYS Mechanical

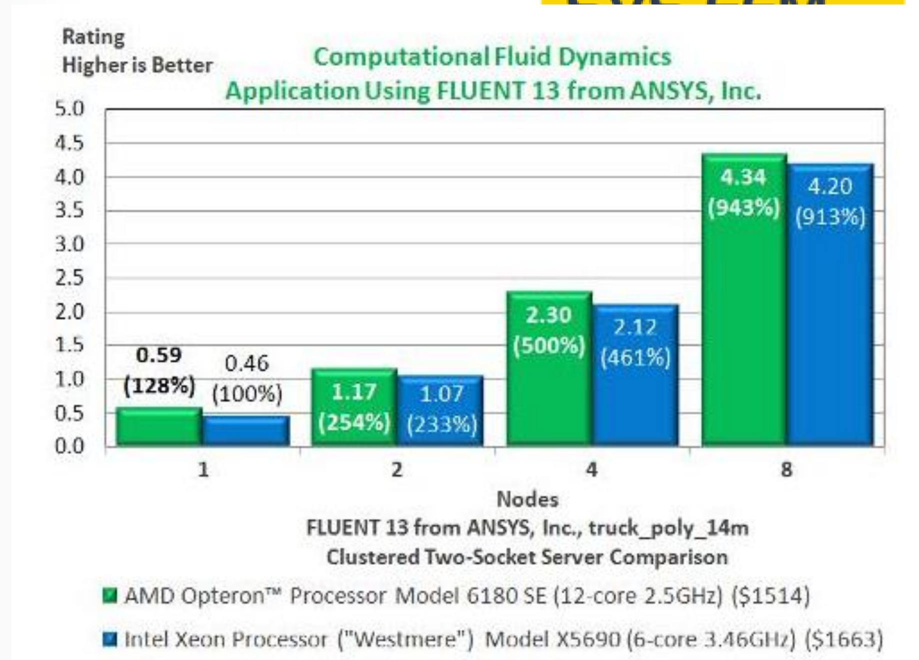
- **In our tests we've seen only a 2% increase in performance although you can see the actual performance can be quite variable .**
- **It is worth noting that this has licensing implications as you would need to oversubscribe the physical cores and hence would need double the HPC Licenses.**

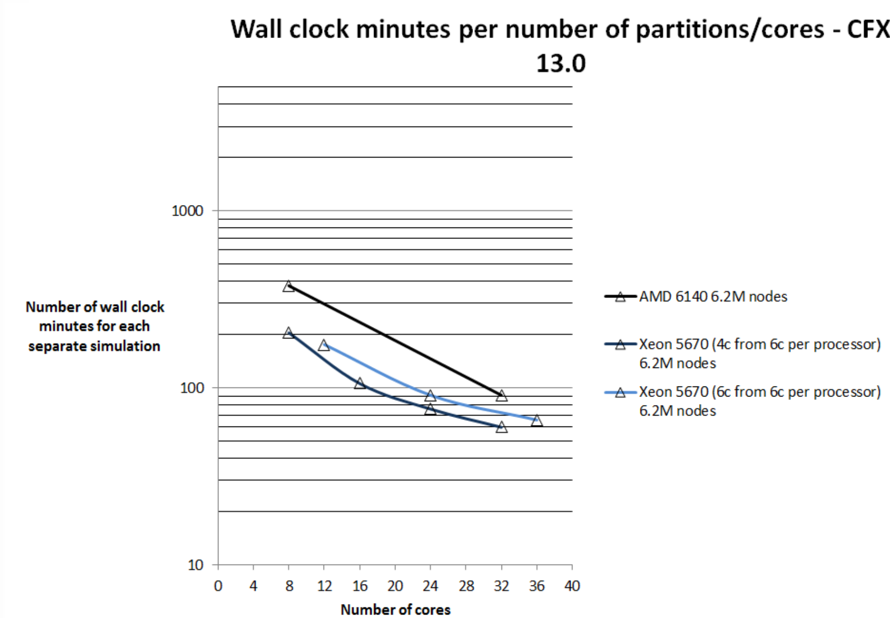


Hyper-threading is NOT recommended

AMD vs. Intel

- These are the published figures from v13.0 for FLUENT. Notice the results here are per node not per core.
- In this case this for the 8 node job this is 192 cores vs. 96 cores. It does not state if SMT (hyper-threading) is used here.
- Below is a CFX job run on the slightly slower AMD 6140 and in comparison to the 5670 (2.93GHz 5.8 GT/s QPI rating). It also only has 8 cores instead of the 12 the 6180 has.
- We can see that the AMD 6140 requires a 1/3 again the number of cores to keep pace with the Intel x5670.
- This is consistent with the above FLUENT results.





Comparing 2 socket vs. 4 socket systems

- **Current 4 socket systems come up slower than their 2 socket counterparts (based on Intel Westmere vs. Xeon E7-8837).**
  - **Clock speed slower**
  - **Memory speed slower**
  - **No additional memory bandwidth.**

Performance of ANSYS Fluent on two-socket and four-socket based systems Performance measure is Fluent Rating (higher values are better)							
2-socket based Systems IBM HS22/HS22V Blade, 3550/3650 M3, Dx360 M3 (Xeon 5600 Series)				4-socket based Systems IBM HX5 Blade, X3850 (Xeon E7-8837 series)			
Nodes	**Sockets**	**Cores**	**Fluent Rating**	**Nodes**	**Sockets**	**Cores**	**Fluent Rating**
1	2	12	88	1	2	16	96
2	4	24	173	1	4	32	188

Comparing 1 node vs. 2 node for 4 socket system

- **Comparison with equal number of cores between:**

  Locally distributed solution (all cores on the same mother board)

  Distributed solution over 2 nodes (with a 1 Gbit network)

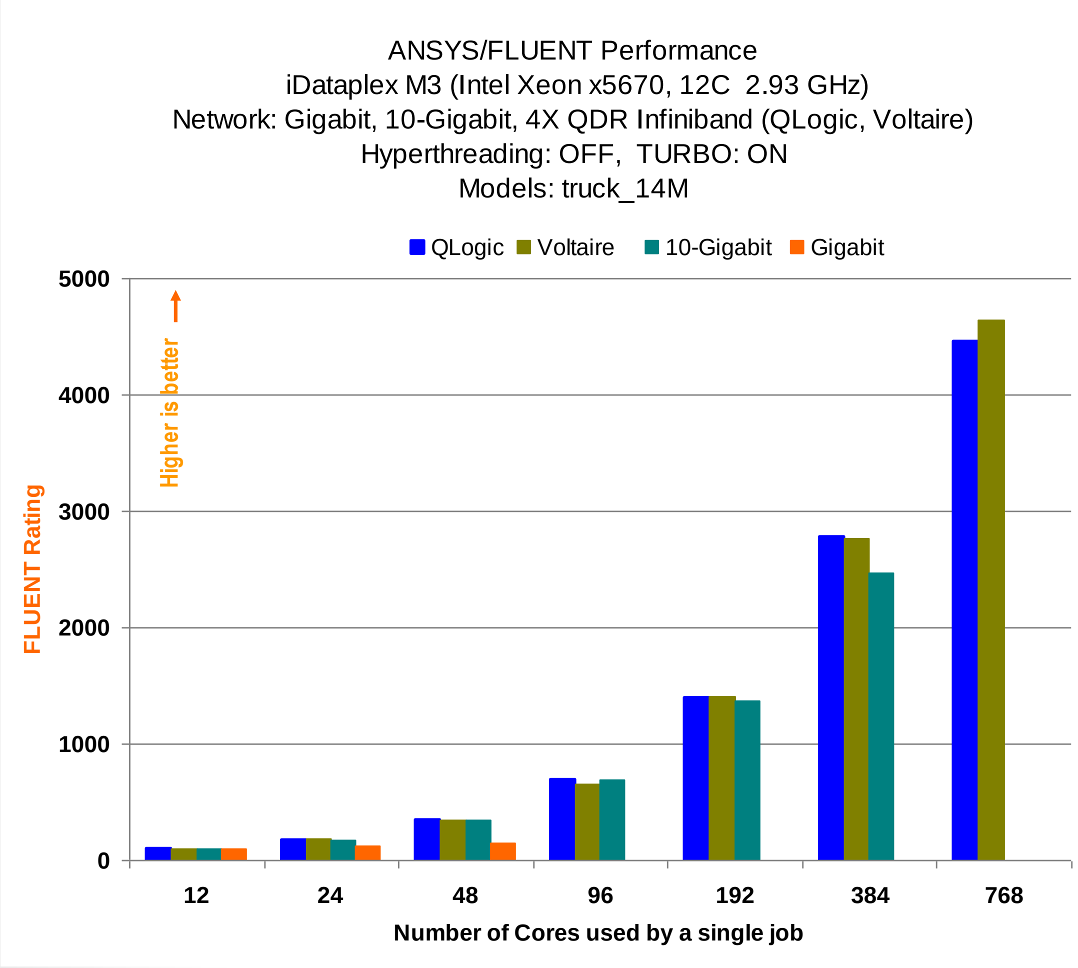
Performance of ANSYS Mechanical on one node and two node based systems Performance measure is elapsed time (lower values are better)							
1 Node 4-socket based Systems Dell R910 4 socket X7560 @ 2.26 GHZ 256 GB RAM				2 Node 4-socket based Systems Dell R910 4 socket X7560 @ 2.26 GHZ 256 GB RAM			
M DOF	**Solver**	**Cores**	**Elapsed Time**	**M DOF**	**Solver**	**Cores**	**Elapsed Time**
12.3	PCG	12	1159	12.3	PCG	6+6	952
1.54	Sparse	12	1595	1.54	Sparse	6+6	1558

Conclusion with a same number of core :

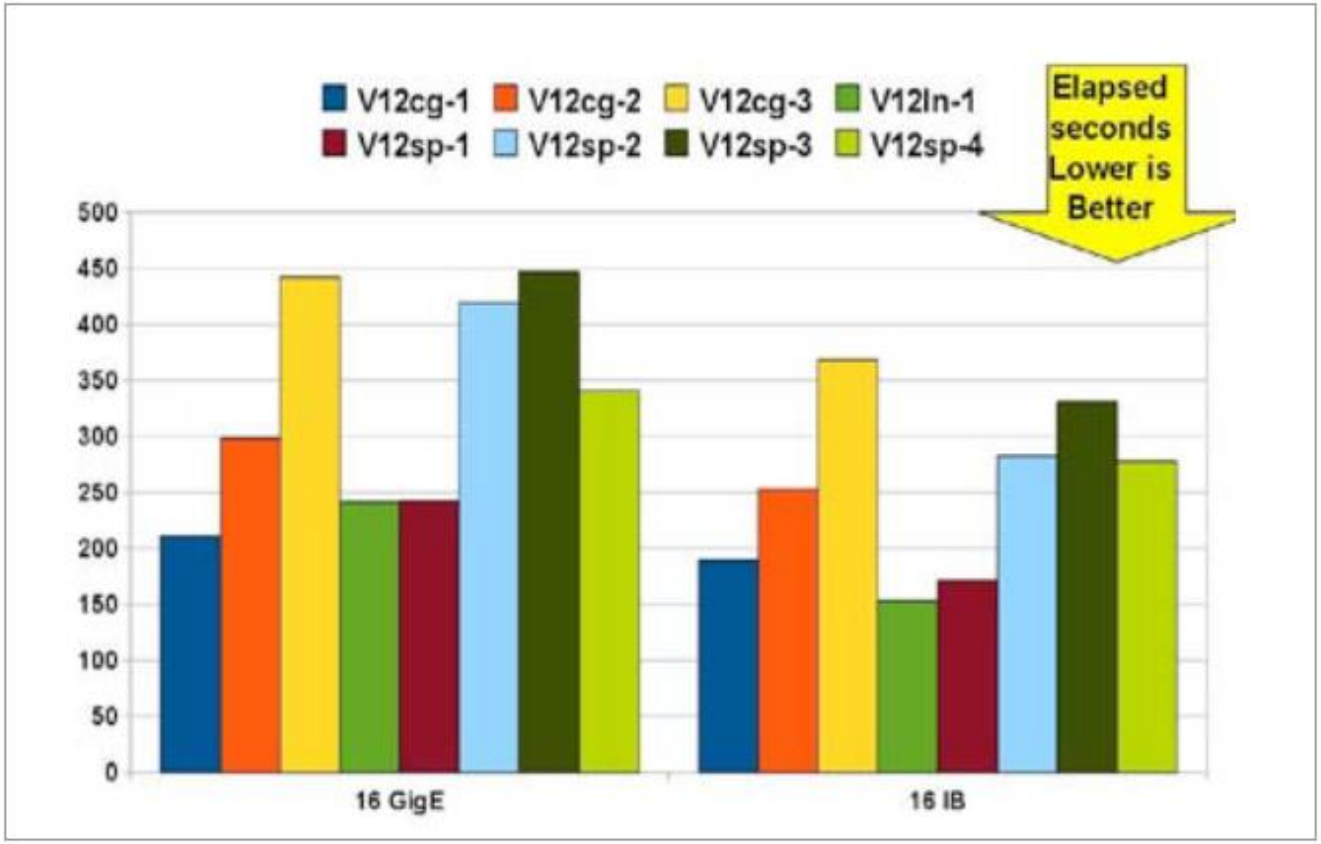
- **PCG solver works better on multiple-node configuration**
- **Performance of sparse solver is less sensitive for multiple-node configuration**

Understanding the effect of the interconnect

- When going for multiple systems linked together the interconnect becomes an important factor.
- The interconnect is the fabric that connects the nodes.
- We can see from the graph opposite with FLUENT how quickly the performance of Gigabit Ethernet drops off.



Understanding the effect of interconnect bandwidth



- Effect of interconnect bandwidth on the DMP benchmarks running with 16 MPI processes across 2 nodes
- Higher bandwidth most relevant for sparse solver benchmarks

Disks/Storage

- IDE, SCSI, SAS, SSD

Faster

Převažuje-li Sparse/Block Lanczos/out-of-core > SAS & SSD v RAID0

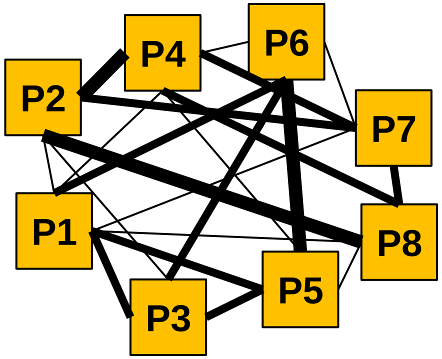
- RAID setups
- RAID 0 – for speed
- RAID 1,5 – for redundancy

- Parallel File Systems
- Only required for large clusters

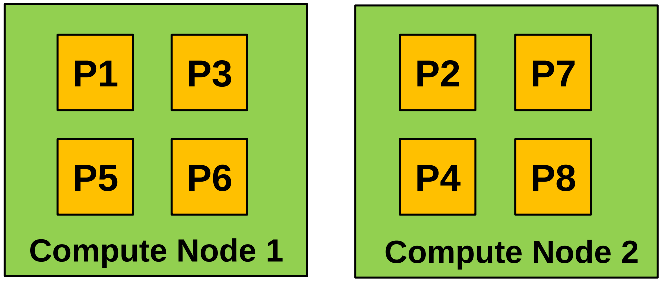
  November 25, 2013

ANSYS CFX Partitioning

- Optimize parallel partitioning in multi-core clusters (CFX)$^{\beta}$
- Partitioner determines number of connections between partitions and optimizes part.-host assignments

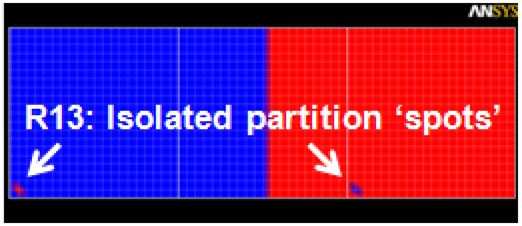


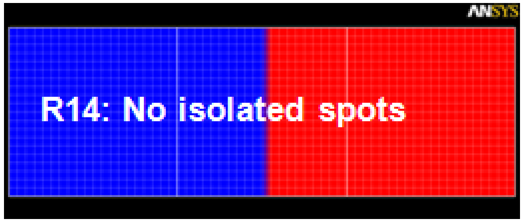
Partitioning step finds adjacency amongst partitions; partitions with max adjacency are grouped on same compute nodes



- Re-use previous results to initialize calculations on large problem (CFX) $^{\beta}$
- Large case interpolation for cases with >~100M nodes

- Clean up of coupled partitioning option for multi-domain cases (CFX)
- Eliminates 'isolated' partition spots





Dramatically reduced partitioning times for cases with fluid-solid interfaces *and* very large numbers of regions

Fluids I/O

FLUENT, CFX and AUTODYN use a “singular” file structure.

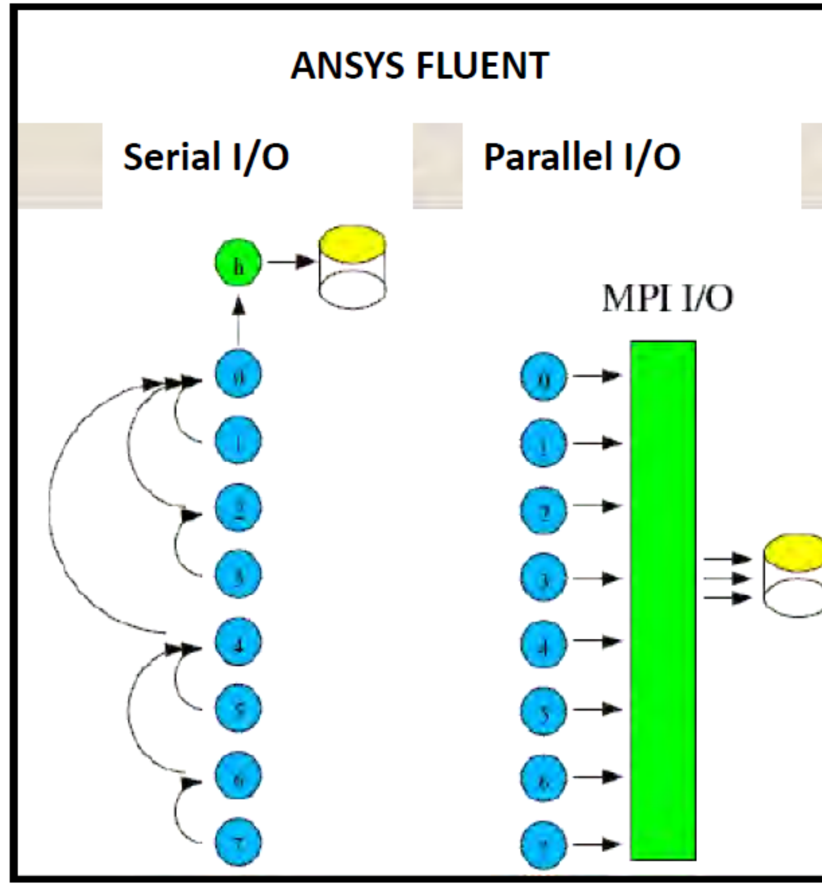
- This means there is one global set of files and every process writes to them.

This methodology falls down at a large number of cores where the file I/O becomes a bottleneck.

- CFX deals with this by using inline compression (cdat)
- FLUENT has both inline compression (cdat) and at v12.x introduced support for a Parallel File (pdat).

Parallel file system support in ANSYS FLUENT

  – ~10x - 20x speedup for data write
  – Eliminates scaling bottleneck for data intensive simulations on large clusters (e.g., transient flows)



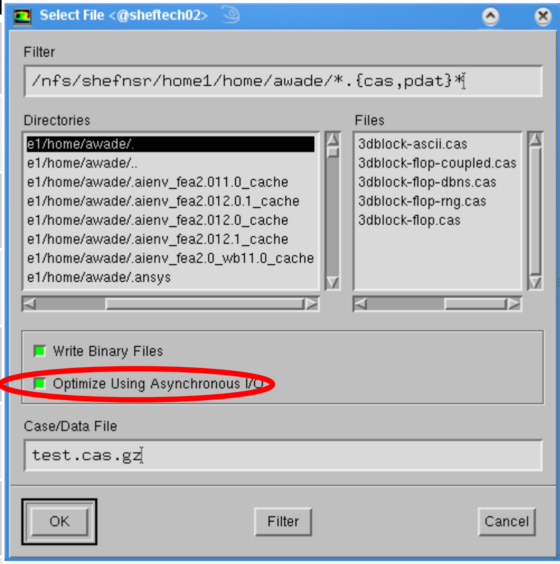
Fluids I/O

Asynchronous I/O for Linux Fluent

Total write time 3-5x quicker over NFS

Even larger speed-ups on bigger cases and local disk (up to 10x)

Mesh	File	Location	Async I/O	Time
15M	Cas	NFS	OFF	217s
15M	Cas	NFS	ON	62s
15M	Dat	NFS	OFF	113s
15M	Dat	NFS	ON	8s
30M	Cas	NFS	OFF	207s
30M	Cas	NFS	ON	75s
30M	Dat	NFS	OFF	144s
30M	Dat	NFS	ON	10s



  November 25, 2013

Optimální nastavení úlohy pro HPC FEM

Mechanical APDL // Performance Guide

SPARSE: obecně 3D, bázové fce vyššího stupně > vhodnější pro GPU (vyjímka modální analýza)

PCG: Low Level difficulty > vhodnější pro GPU

SMP: max do 8 jader (nejvyšší efektivita při výpočtu na 4)

DMP: u FEM nejefektivejší do 32 jader, použitelné maximum 128, u CFD dobrá škálovatelnost až do 1024 jader, ale lze použít až do 3072 jader

In-core: optimální režim výpočtu v RAM (oproti out-of-core = swap file). In-core lze vynutit správnou alokací paměti

`Ansys145 -m ### -db ###`

Nebo pomocí příkazu: `BCSOPTION,,INCORE` příp. `DSPOPTION,,INCORE`

Pro FEM maximalizace I/O -> RAID0 + >4x SSD/SAS, propustnost >500 MB/s, lze si pomoci s RAMDiskem (linux). U clusteru je vhodné lokální I/O storage pro každé PC.

Pro CFD maximalizace síťové propustnosti, propustnosti RAM/bus, fragmentace sítě, nastavení MPI

RAM (in-core, FEM):

SPARSE solver: SMP → 10GB/1mil DOFs, DMP → 10GB/1mil DOFs * 1/#cores (BCSoption, DSPoption)
PCG solver: SMP → 1GB/1mil DOFs, DMP → 2GB/1mil DOFs * 1/#cores (PCGOPT,LevelDiff; MSAVE,on) (vyšší Level → více RAM/rychlejší)

Block-Lanczos: SMP → 15-20GB/1mil DOFs (MODOPTION,BlockSize) (větší blok → více RAM/rychlejší)
PCG-Lanczos: SMP → 2GB/1mil DOFs, DMP → 3GB/1mil DOFs * 1/#cores (PCGOPT,5; MSAVE,on) optimální pro výpočet přes více uzlů v non-shared mem.
Supernode: SMP → mezi Block a PCG Lanczos (SNOPTION,BlockSize)

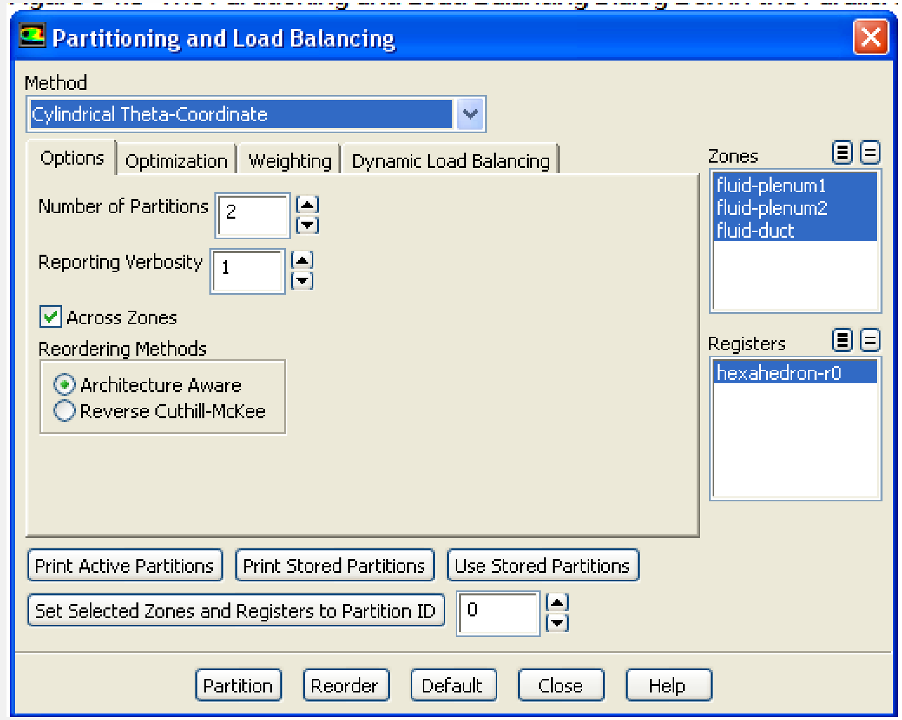
Kontakty: redukce pomocí CNCHECK,TRIM

Redukce při spojování souborů po DMP výpočtu: DMPOPTION (specifikuje, které soubory se mají spojit – RSM, EMAT, ESAV,...), COMBINE/RESCOMBINE

Metody doménového dělení: DDOPTION,GREEDY/METIS

Optimální nastavení úlohy pro HPC CFD (Fluent)

- **Nastavení dělení domény:**
  - **Parallel → Auto Partition (15 metod)**
  - **Parallel → Partitioning and Load Balancing... (manuální)**



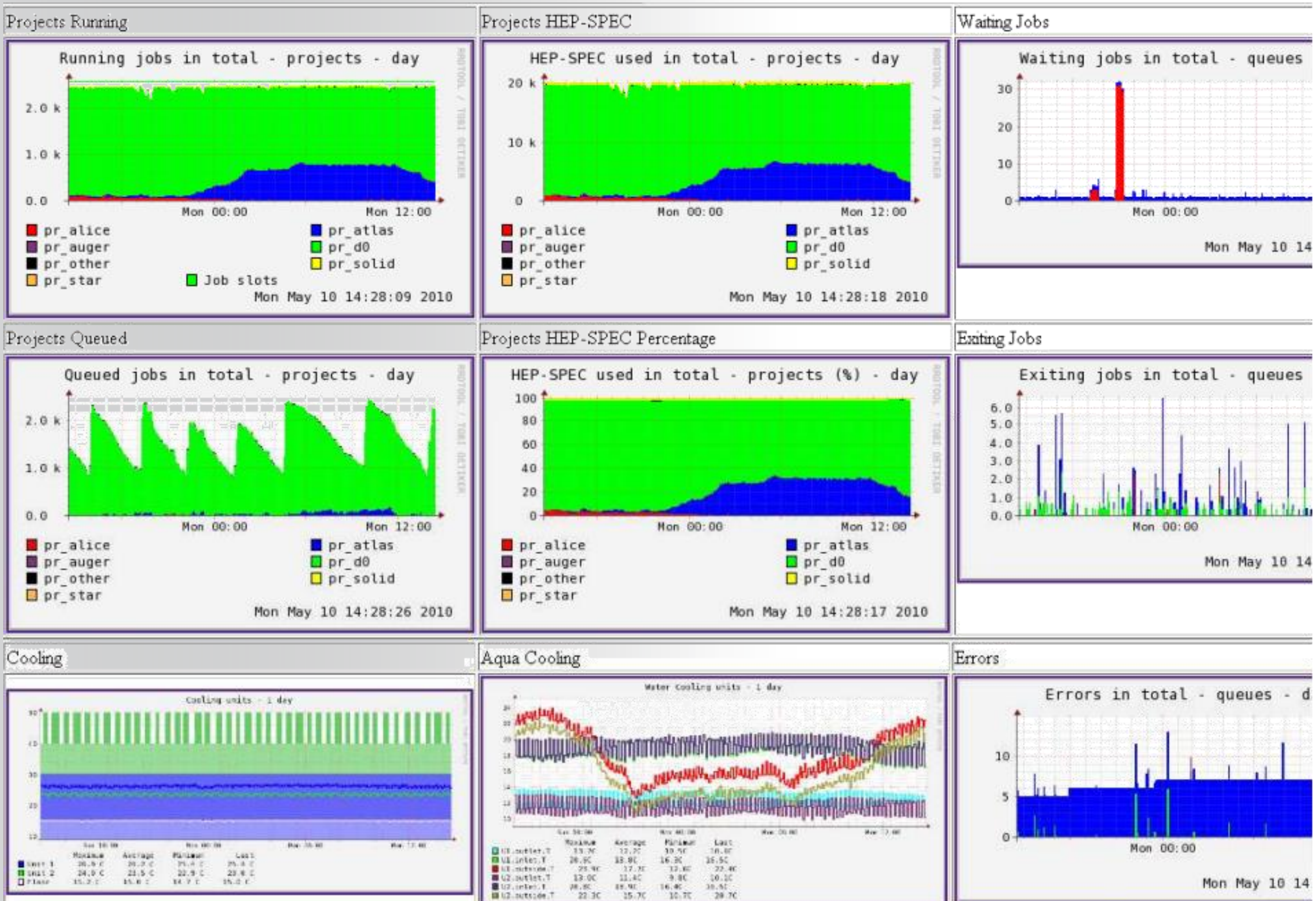
Optimální nastavení úlohy pro HPC CFD (CFX)

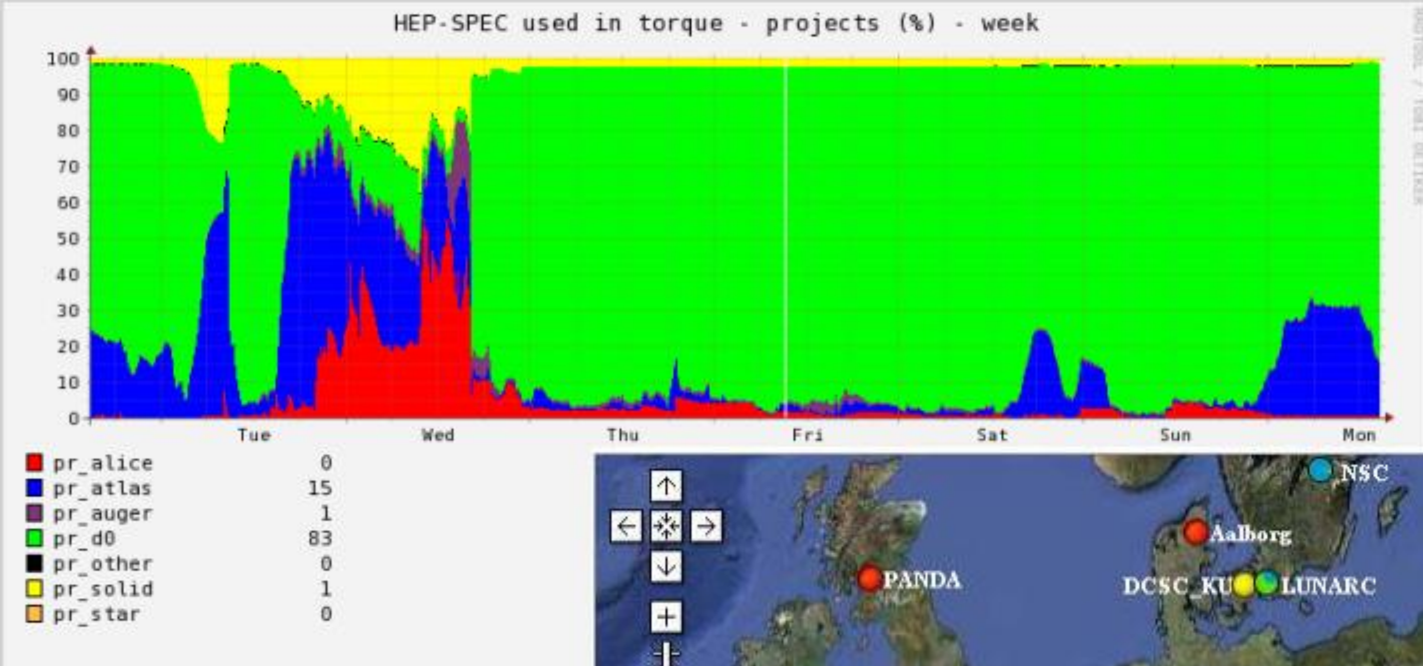
- Pro řešení jedné úlohy se snažte používat stroje s podobným výkonem
- Vyhněte se používání swapovacího souboru
- Používejte rychlou síťovou konektivitu (1Gbps/Infiniband)
- Inifiband je podporován pouze přes Platform-MPI
- Před spuštěním úlohy zajistěte dostupnost uzlů (nesmí být sdílené jinou úlohou)
- Vyhněte se přidělování většího počtu vláken než je počet jader
- Používejte „rozumný“ počet partitions. Orientačně počet part.~=počet jader

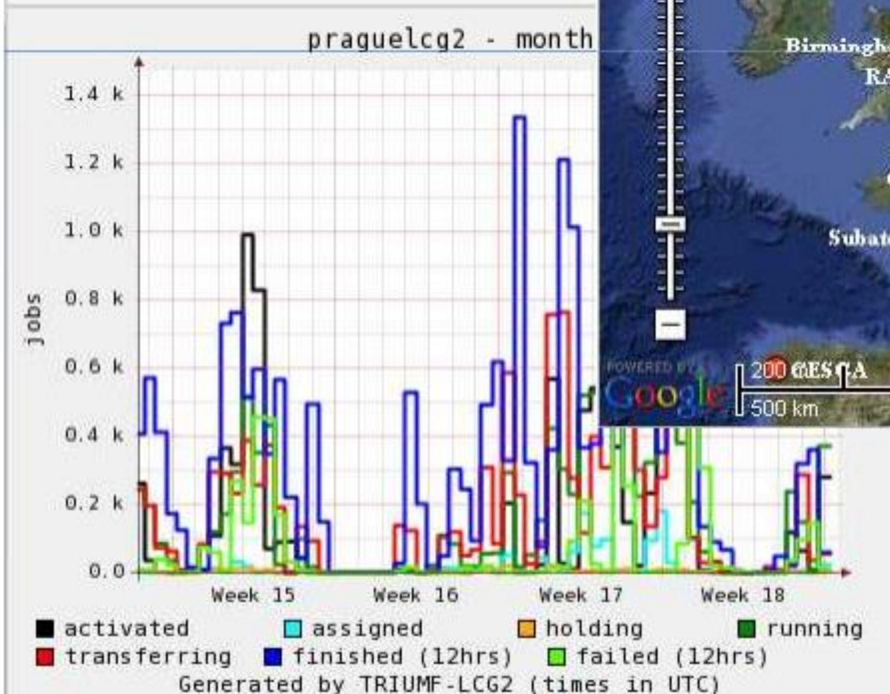
ANSYS

Monitoring-Ganglia

SVS FEM
Your partner in computing




HEP-SPEC used in torque - projects (%) - week
100
90
80
70
60
50
40
30
20
10
0
Tue
Wed
Thu
Fri
Sat
Sun
Mon
pr_alice 0
pr_atlas 15
pr_auger 1
pr_d0 83
pr_other 0
pr_solid 1
pr_star 0

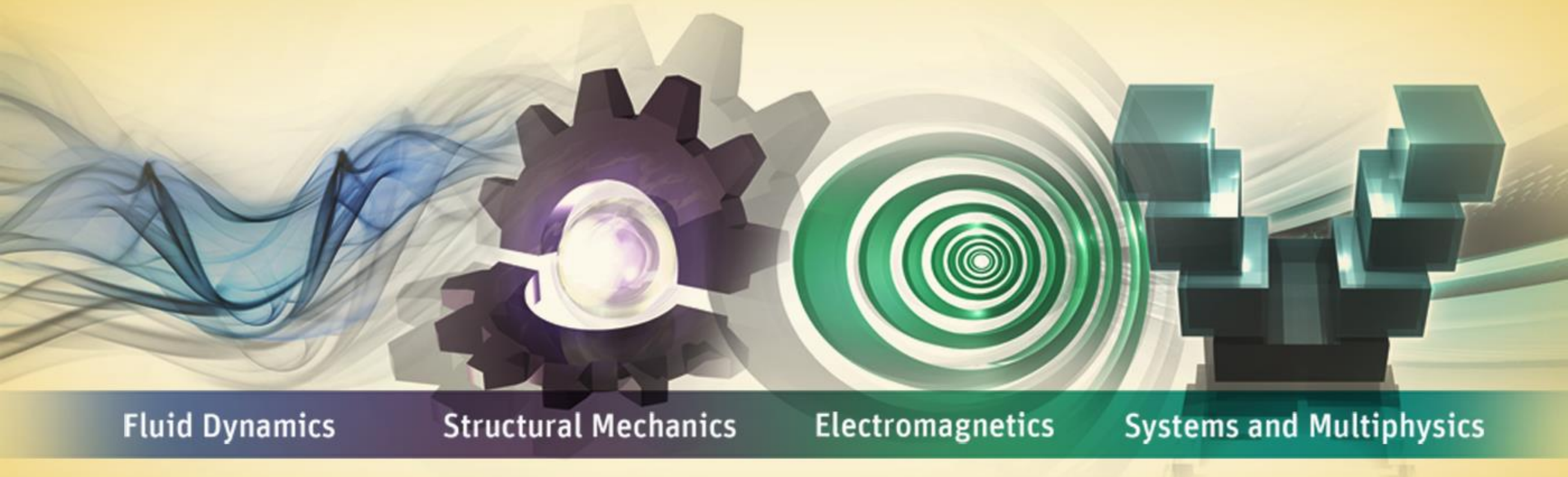

praguelcg2 - month
jobs
1.4 k
1.2 k
1.0 k
0.8 k
0.6 k
0.4 k
0.2 k
0.0
Week 15
Week 16
Week 17
Week 18
activated
assigned
holding
running
transferring
finished (12hrs)
failed (12hrs)
Generated by TRIUMF-LCG2 (times in UTC)


Map
Satellite
Hybrid
NSC
Aalborg
JINR
RRC-KI
Troitsk
ITEP
PANDA
DCSC_KU
LUNARC
IHEP
Birmingham
NIKHEF
SARA
Poznan
WUT
RAL
Muenster
Dortmund
Prague-CREAM
KPI
BITP
KNU
IC
ISMA
GSI
Cyfronet
GRIF_IRFU
GRIF_IPNO
FZK
Strasbourg_IRES
Kosice
Bratislava
KFKI
Subatech
CERN
Clermont
IPNL
CCIN2P3
Torino-CREAM
Legnaro
CNAF
ISS
NIHAM
Grenoble
Bologna
Florence
500 km
Imagery ©2010 TerraMetrics - Terms of Use

Realize Your Product Promise™

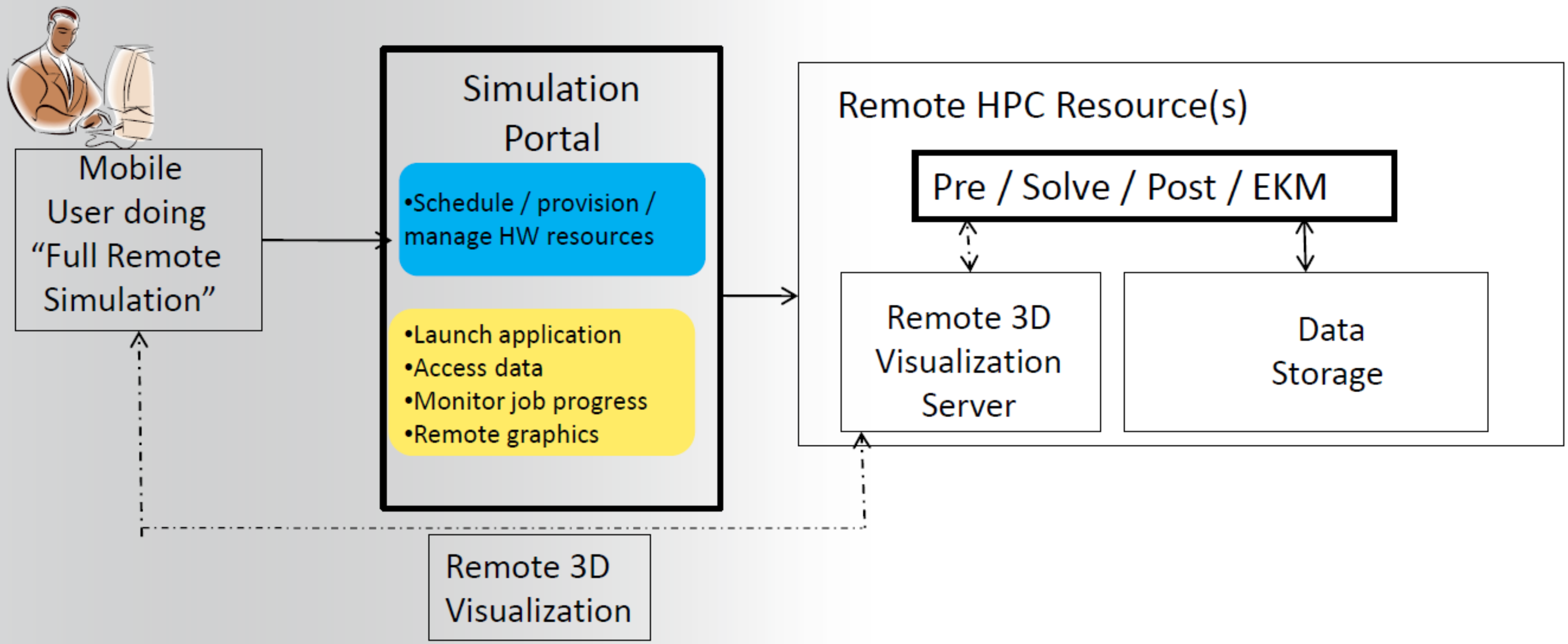
ANSYS Cloud

NICE DCV, EngineFrame, Vcollab, EKM



SVS FEM
Your partner in computing

Interaktivní vzdálený přístup - Cloud



- **Soubory jsou výhradně jen na HPC prvcích** – efektivní a pokročilá správa dat, spolupráce
- **Celá simulace probíhá zcela ve vzdáleném režimu (Pre/Solv/Post)**
- **Dostupná a plně implementovatelná technologie současnosti** (u těch nejnáročnějších)

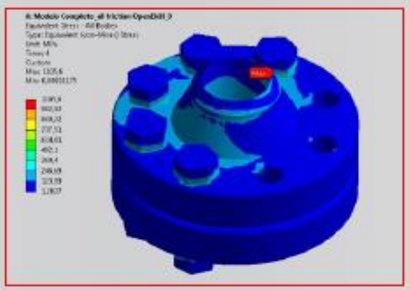
Simulace online

An automated web-based application for field engineers

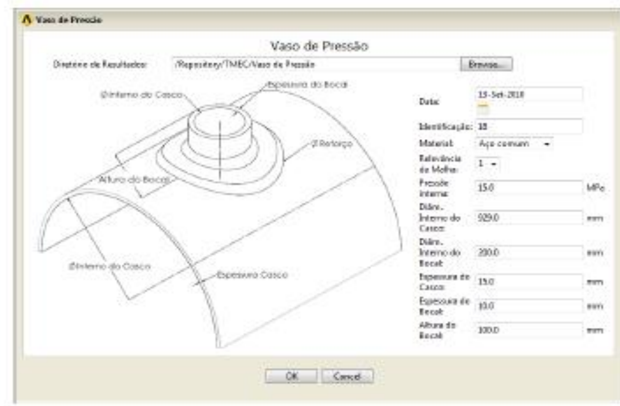
1) Open web-browser

2) Login into EKM

3) Open the Application and Request the Analysis



- *Select the Equipment*
- *Write a Case Description*
- *Set Material*
- *Set Dimensions*
- *Set Operating Conditions*
- *Define Design Restrictions*
- *...*
- *Request advanced analysis to CENPES*

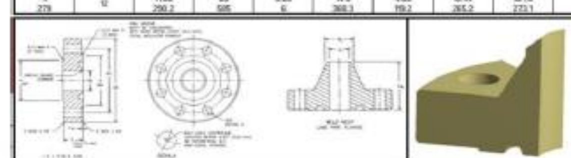


4) Receive E-mail notification with the link to the HTML Report and Simulation Results

5) Open the report in the browser and analyze the results and make decisions.

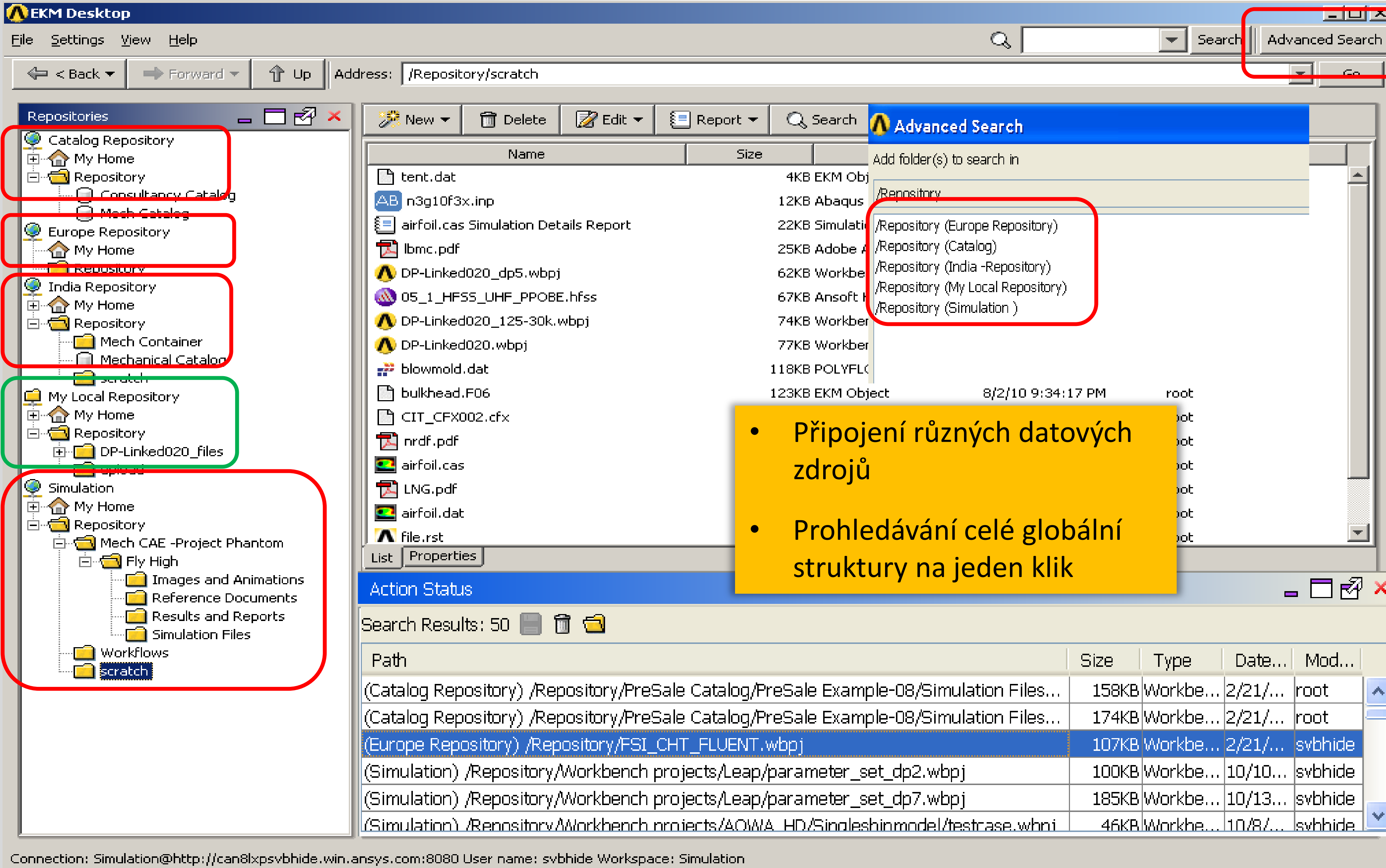
Flange, Hub and Bore Dimensions

Nominal Diameter	Number of bolts	Max Bore	Outside Diameter of Flange	Max Chamfer	Diameter of Hub	Total Thickness of Flange	Hub length Welding-Neck Line-Pipe Flange	Neck Diameter Welding-Neck Line-	Slice Angle	Maximum Bore of Welding Neck Flange	BX Ring Number
		B	OD	C	X	T	L_a	H_1		J_1	
in (mm)		in (mm)	in (mm)	in (mm)	in (mm)	in (mm)	in (mm)	in (mm)	°	in (mm)	
4 1/16 103	8	4.09 103.9	12.25 310	0.12 3	6.38 162.1	2.44 62	5.19 131.9	4.5 114.3	45	3.44 87.4	155
11 279	12	11.06 280.2	23 585	0.25 6	14.5 368.3	4.69 119.2	10.44 265.2	10.75 273.1	30	8.5 215.9	158



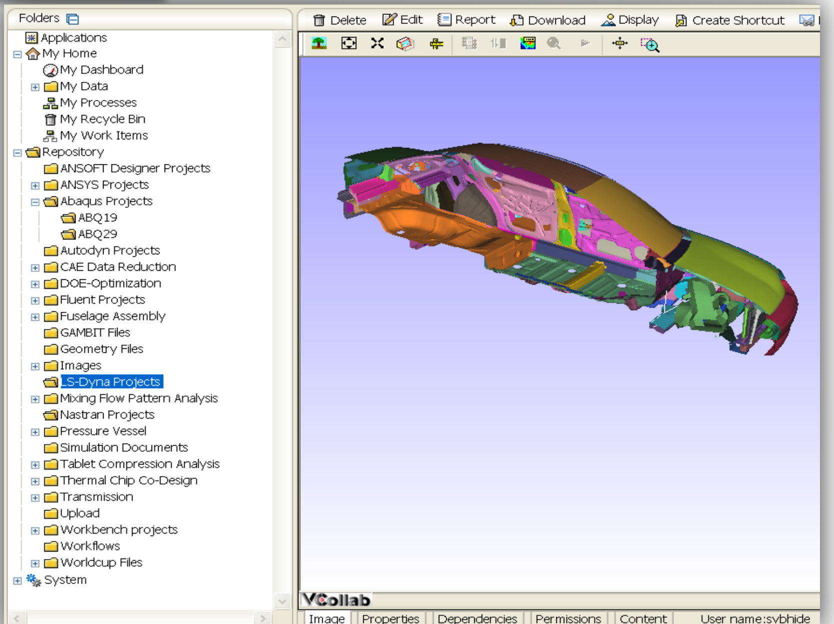
  November 25, 2013

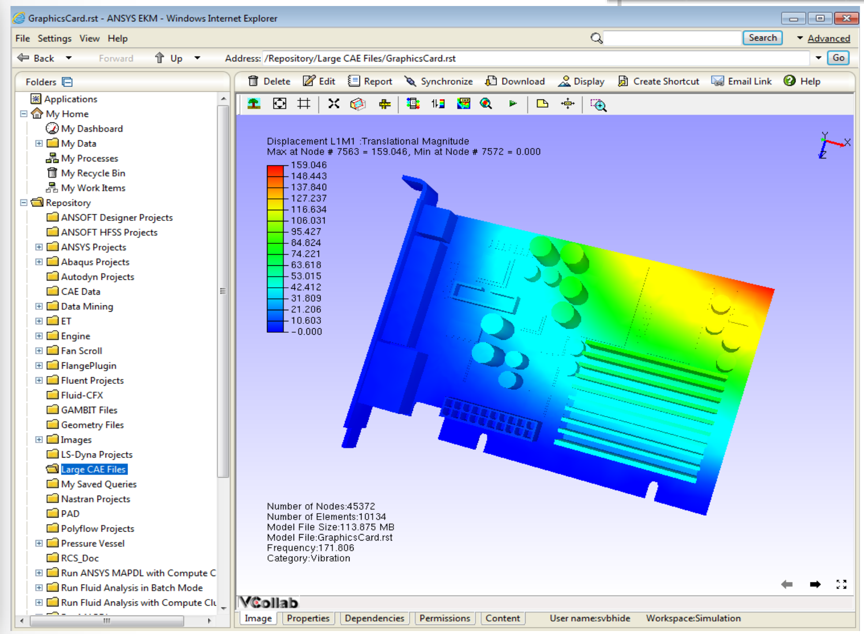
Implementace EKM

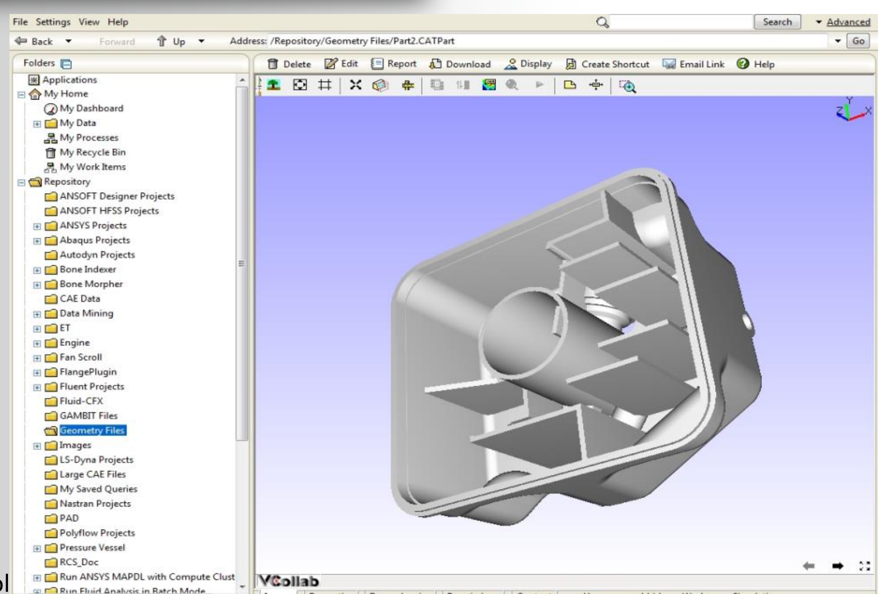


- Připojení různých datových zdrojů
- Prohledávání celé globální struktury na jeden klik

Online vizualizace výsledků



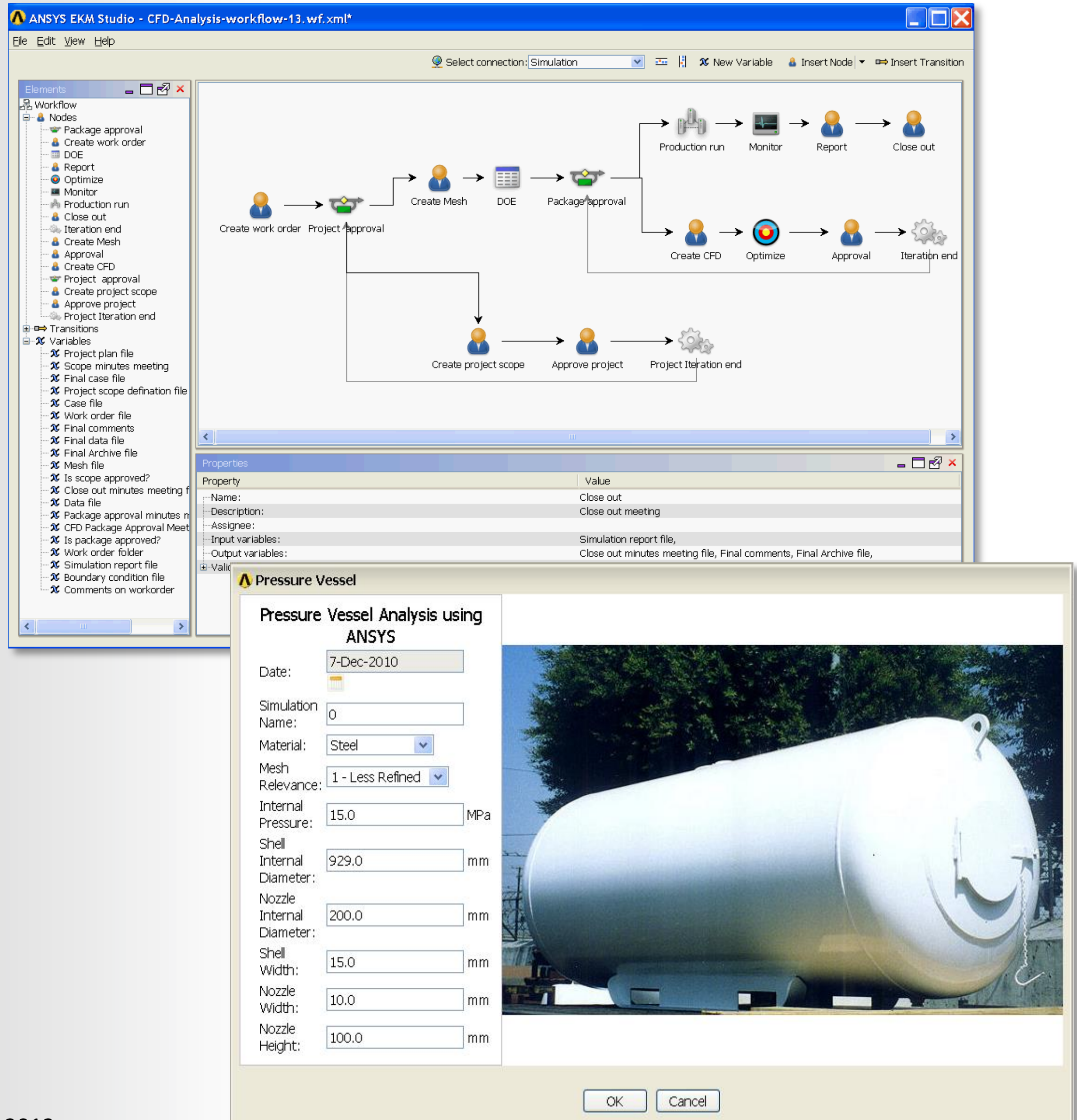




ANSYS Řízení projektu

SVS FEM
Your partner in computing

- Uživatelem definovaná struktura
- Modelování toku výr. procesu
- Šablony (web)
- Možnost vytvářet nadstavby (XML, Python,...)
- Silná podpora metadat dalších aplikací



VCollab

SVS FEM s.r.o. se stala oficiálním partnerem firmy VCOLLAB.

Aplikace VCOLLAB nabízí vizualizaci CAD/CAE dat ANSYSu, ale i SW třetích stran

S. No.	CAD Package	Supported Versions	Supported Extensions
1	Catia V4	4.15 to 4.24	*.model
2	Catia V5	R7 to R20	
3	Unigraphics	v10 to NX7.5	
4	Pro/E	2000 to WildFire5	
5	SolidWorks	1999 to 2011	
6	SolidEdge	Upto ST3	
7	Inventor	10 to 2011	
8	IGES	Upto 5.2	
9	STEP	from AP203/AP2	
10	CGR	R10 to R20	

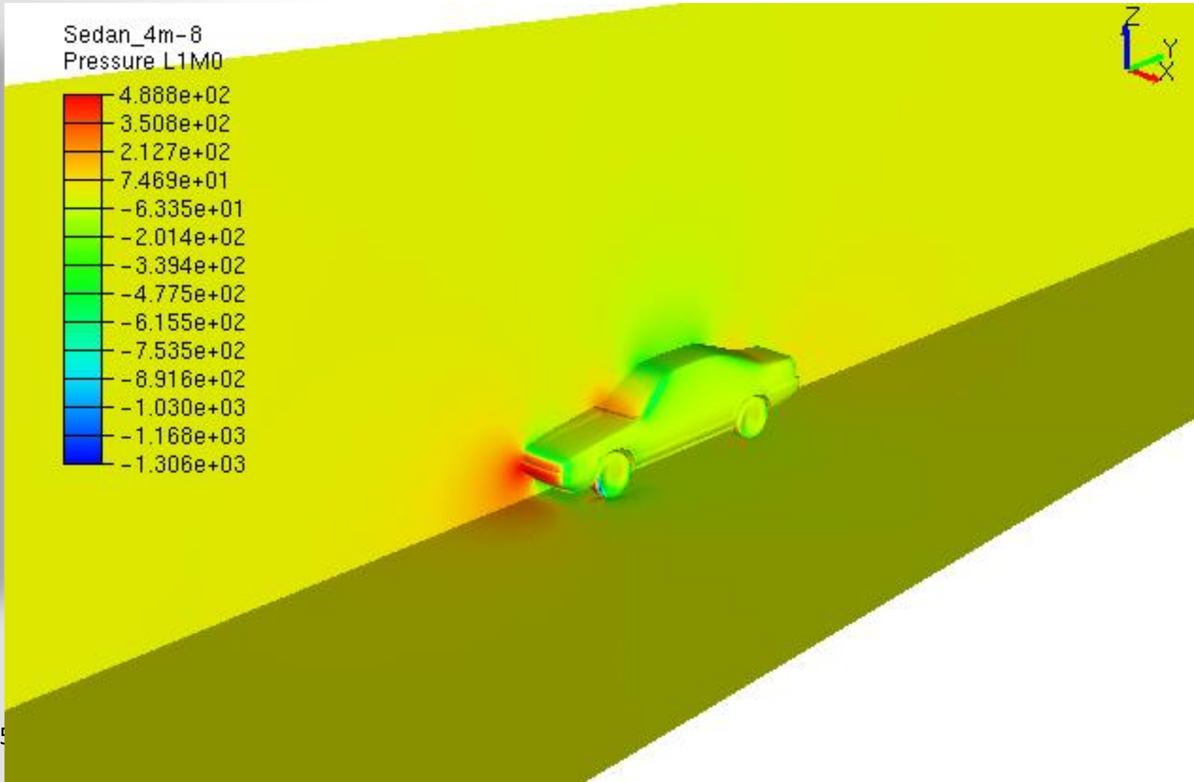
CAE Software	Formats Supported	Remarks
ABAQUS	ODB, FIL, INP	Supports V6.10
MSC/ASTRAN, NX/NASTRAN	OP2, BDF	Supports Complex Eigen Vectors
MSC/MARC	T16 and t17 plot files	
ANSYS	ans,cdb, RST, RTH, RFL	
LS-DYNA	D3PLOT files, key Files	
FLUENT	Binary Output (.CAS and .DAT files)	
ENSIGHT GOLD	.CASE, .ENCASE	
PATRAN	.PAT, .OUT files	
Star-CD (Star-CCM)	.CCM	
FESAFE	.fer	
CFX	.res	
PTC/Mechanica	*.neu	

VCollab vizualizace

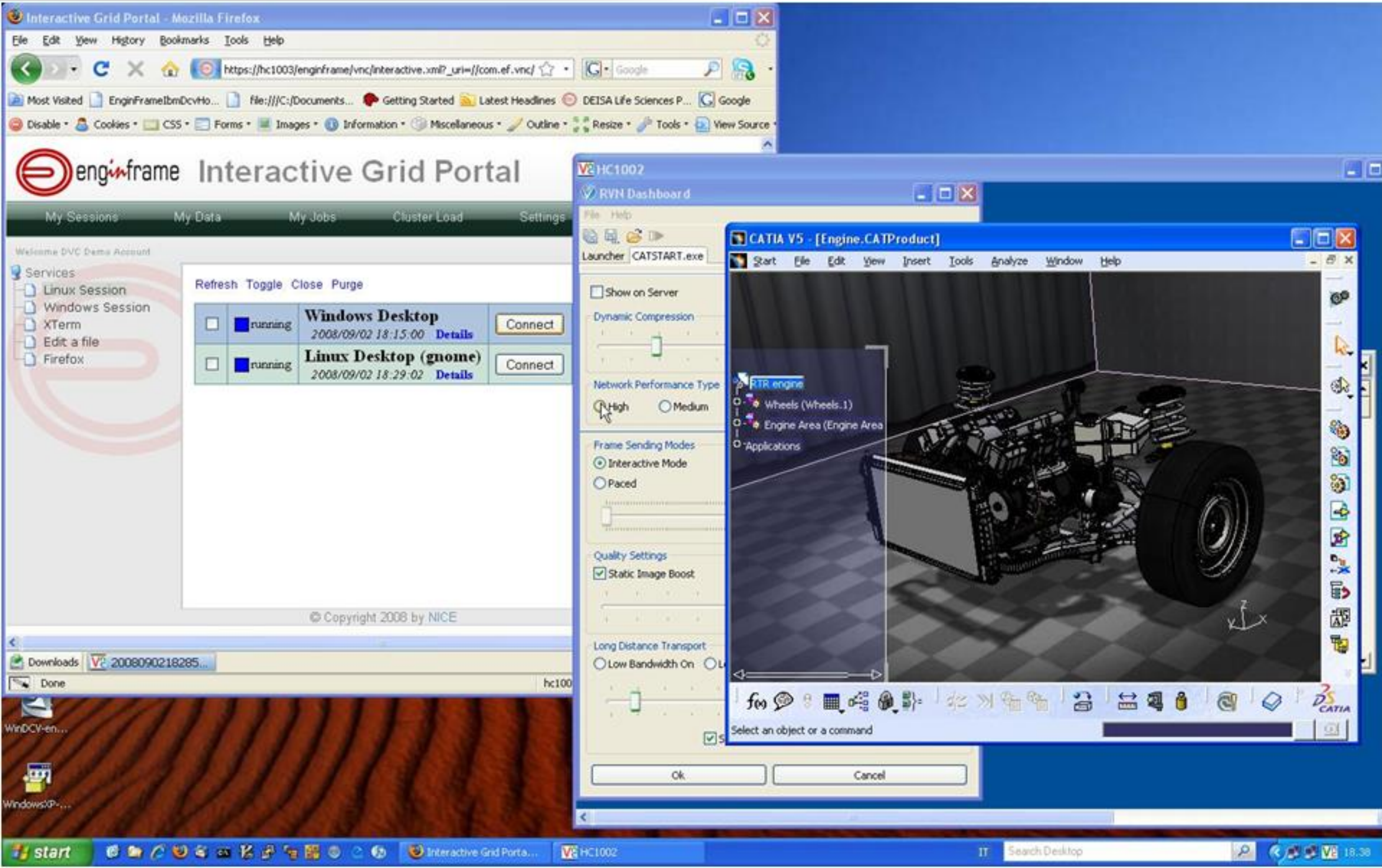
Nabízí velmi propracovaný SW pro převod CAD/CAE dat do vysoce komprimovaného jednotného vizualizačního formátu, který může být poskytován vašim zákazníkům/konzultantům/managerům, bez nutnosti vlastnit vlastní CAE SW.

Výstupy mohou být snadno integrovány do MS PowerPointu, Excelu či prezentovány na webu (viz www.svsfem.cz)

CAE Software	CAE Results File Size (MB)	CAX File Size (MB)	File size Reduction
ABAQUS (FIL)	2,930	47.4	**98%**
MSC NASTRAN	289	46.4	**84%**
MSC MARC	243	23.9	**90%**
ANSYS	14,000	92.0	**99%**
LS DYNA	363	145.0	**60%**
FLUENT	347	13.1	**96%**



NICE DCV

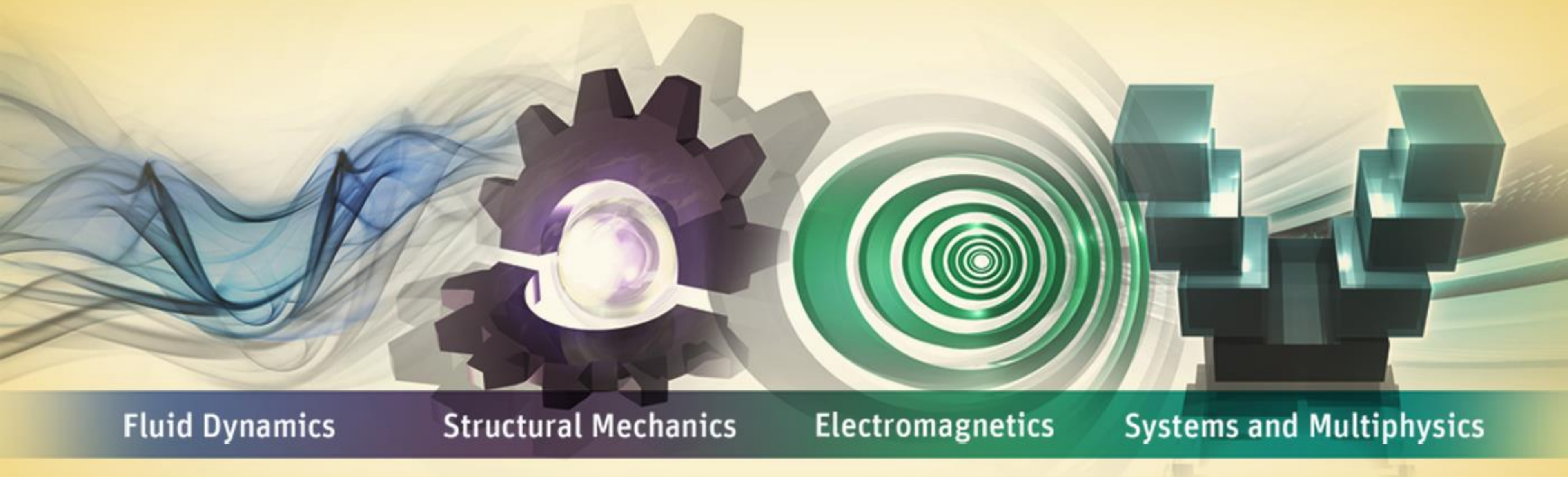


Interactive Grid Portal - Mozilla Firefox
enginframe Interactive Grid Portal
My Sessions
My Data
My Jobs
Cluster Load
Settings
Services
Linux Session
Windows Session
XTerm
Edit a file
Firefox
Refresh Toggle Close Purge
running
Windows Desktop
2008/09/02 18:15:00 Details
Connect
running
Linux Desktop (gnome)
2008/09/02 18:29:02 Details
Connect
© Copyright 2008 by NICE
HC1002
RVN Dashboard
Launcher CATSTART.exe
Show on Server
Dynamic Compression
Network Performance Type
High
Medium
Frame Sending Modes
Interactive Mode
Paced
Quality Settings
Static Image Boost
Long Distance Transport
Low Bandwidth On
Ok
Cancel
CATIA V5 - [Engine.CATProduct]
Start File Edit View Insert Tools Analyze Window Help
RTR engine
Wheels (Wheels.1)
Engine Area (Engine Area
Applications
Select an object or a command
start
Interactive Grid Porta...
HC1002

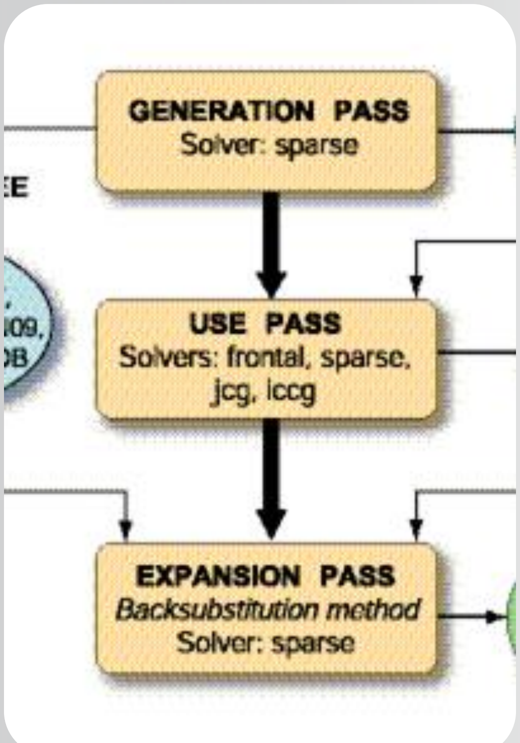
ANSYS

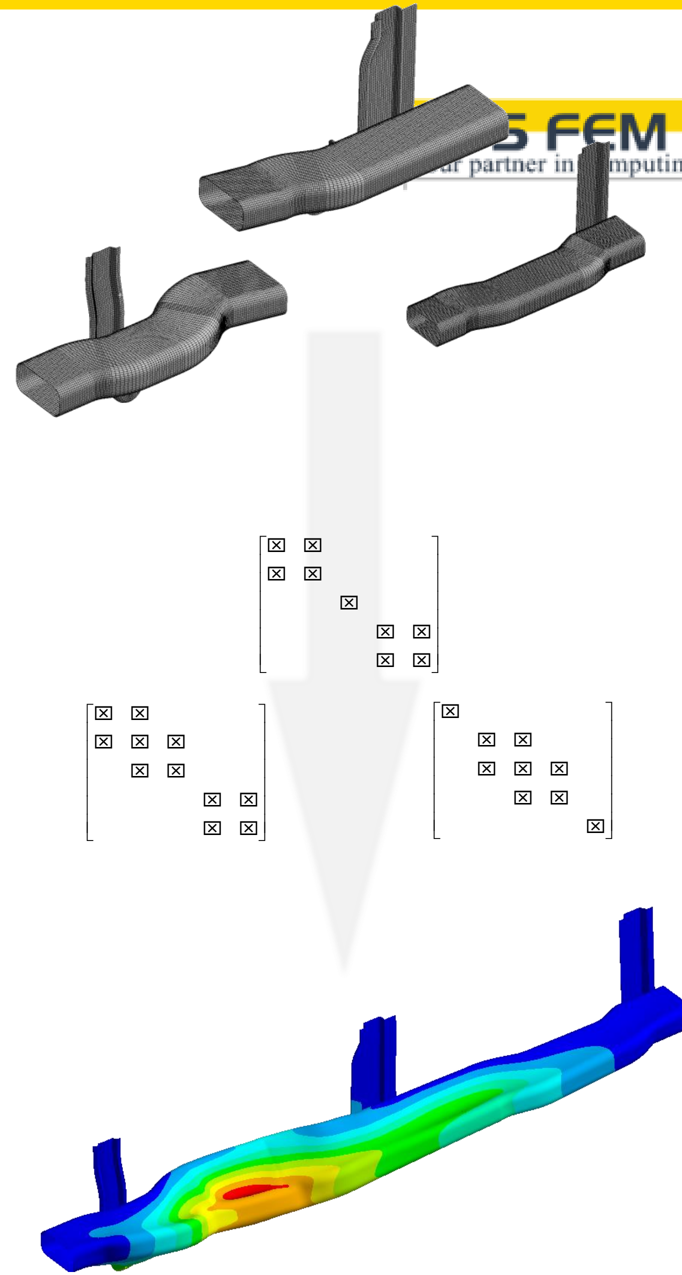
Realize Your Product Promise™

Solution of Large models

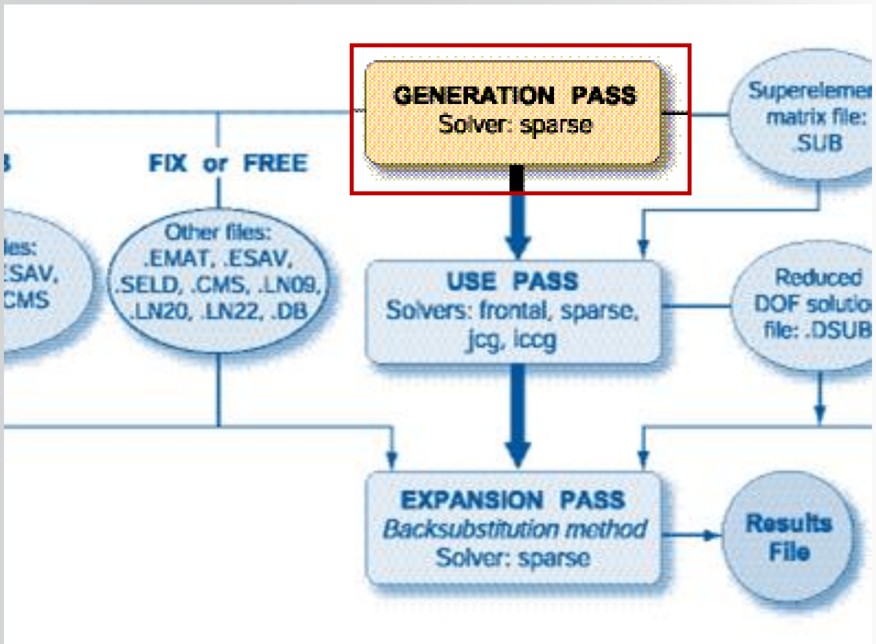


Substructuring allows for collaborative work or memory efficient harmonic and transient simulations

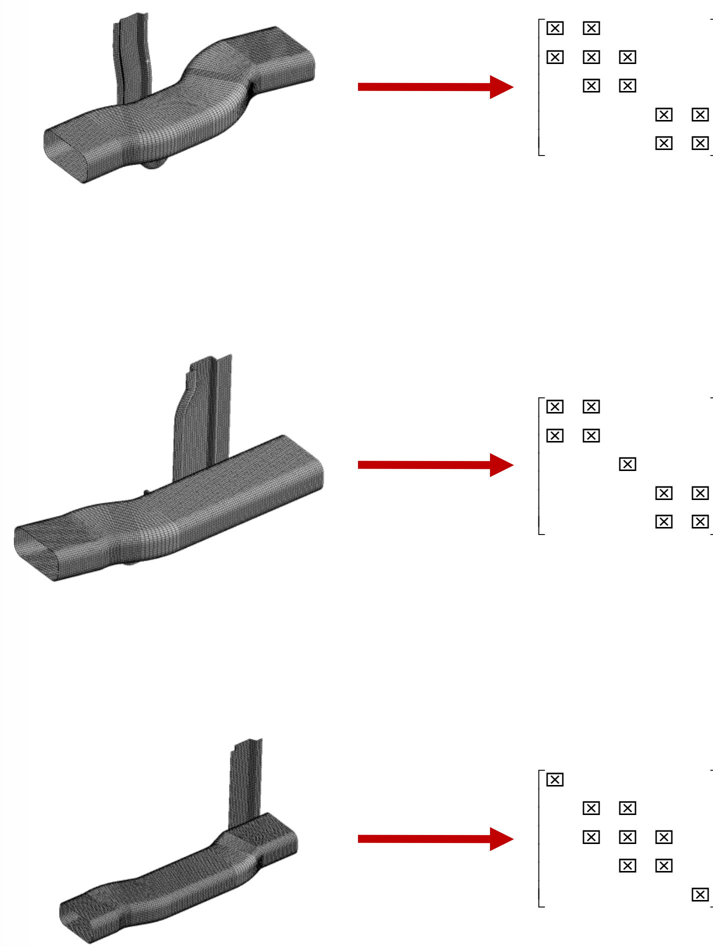




Reduce linear components

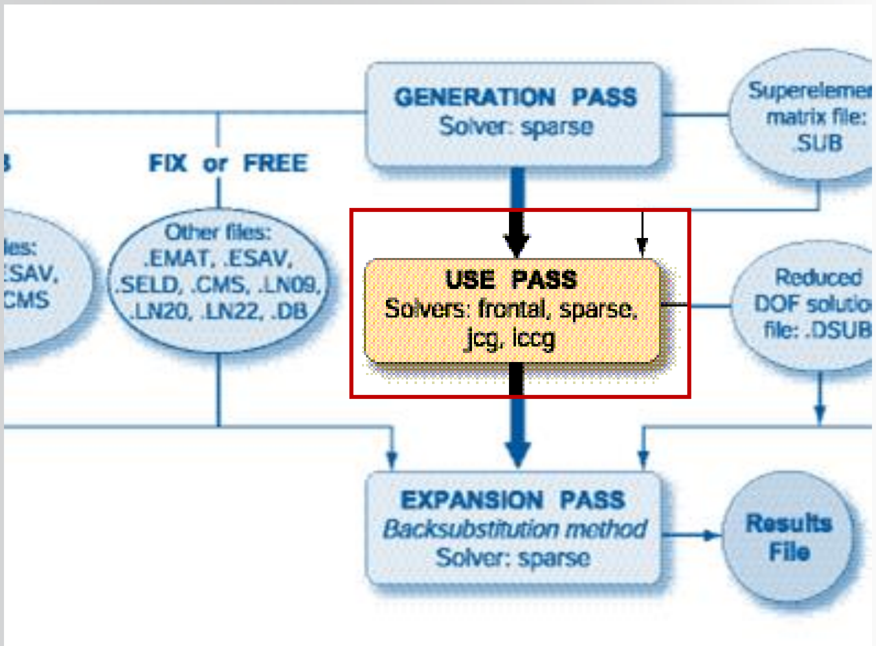


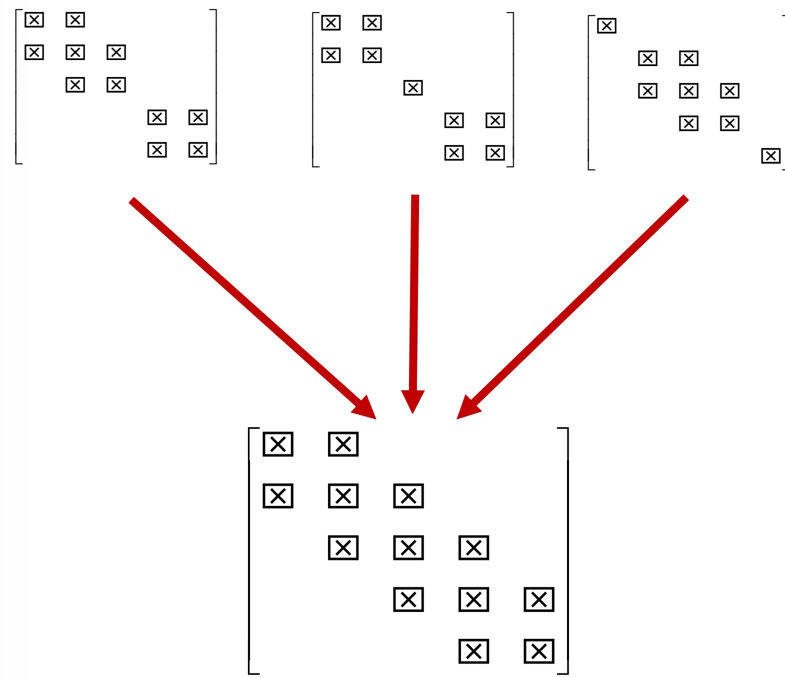
Full components ***Reduced model***



  November 25, 2013

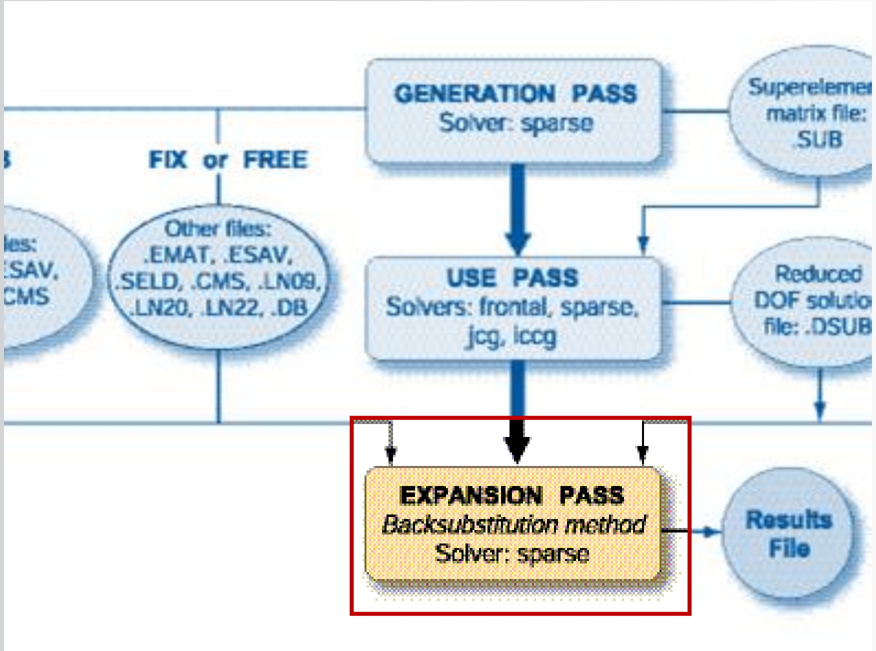
Compute the solution by assembling the reduced models

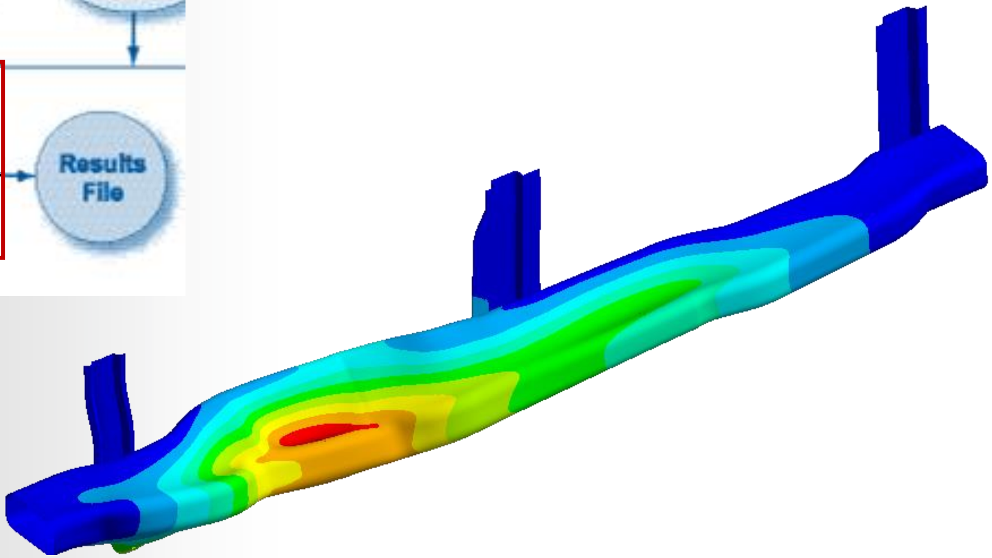




  November 25, 2013

Post-process on full model or areas of interest





Overview – Solution Steps

SVS FEM
Your partner in computing

Generation Pass (Step 1)

SubStructure 1
1. Assemble the M, K
2. Static reduction
3. Modal analysis

SubStructure 2
1. Assemble the M, K
2. Static reduction
3. Modal analysis

…

SubStructure N
1. Assemble the M, K
2. Static reduction
3. Modal analysis

Superelement 1

Superelement 2

…

Superelement N

USE Pass (Step 2)

1.Couple superelements
2. Analysis

Expansion Pass (Step 3)

Transfer back to physical coordinate of substructure 1

Transfer back to physical coordinate of substructure 2

…

Transfer back to physical coordinate of substructure N

Static Substructuring & CMS

- ✓ **Guyan Reduction procedure**
- ✓ **inertia forces are negligible compared to elastic forces**
- ✓ **Net result: the reduced stiffness matrix is exact, whereas the reduced mass and damping matrices are approximate**

Note: Choosing master DOF is an important step in a reduced analysis, impacting accuracy of results

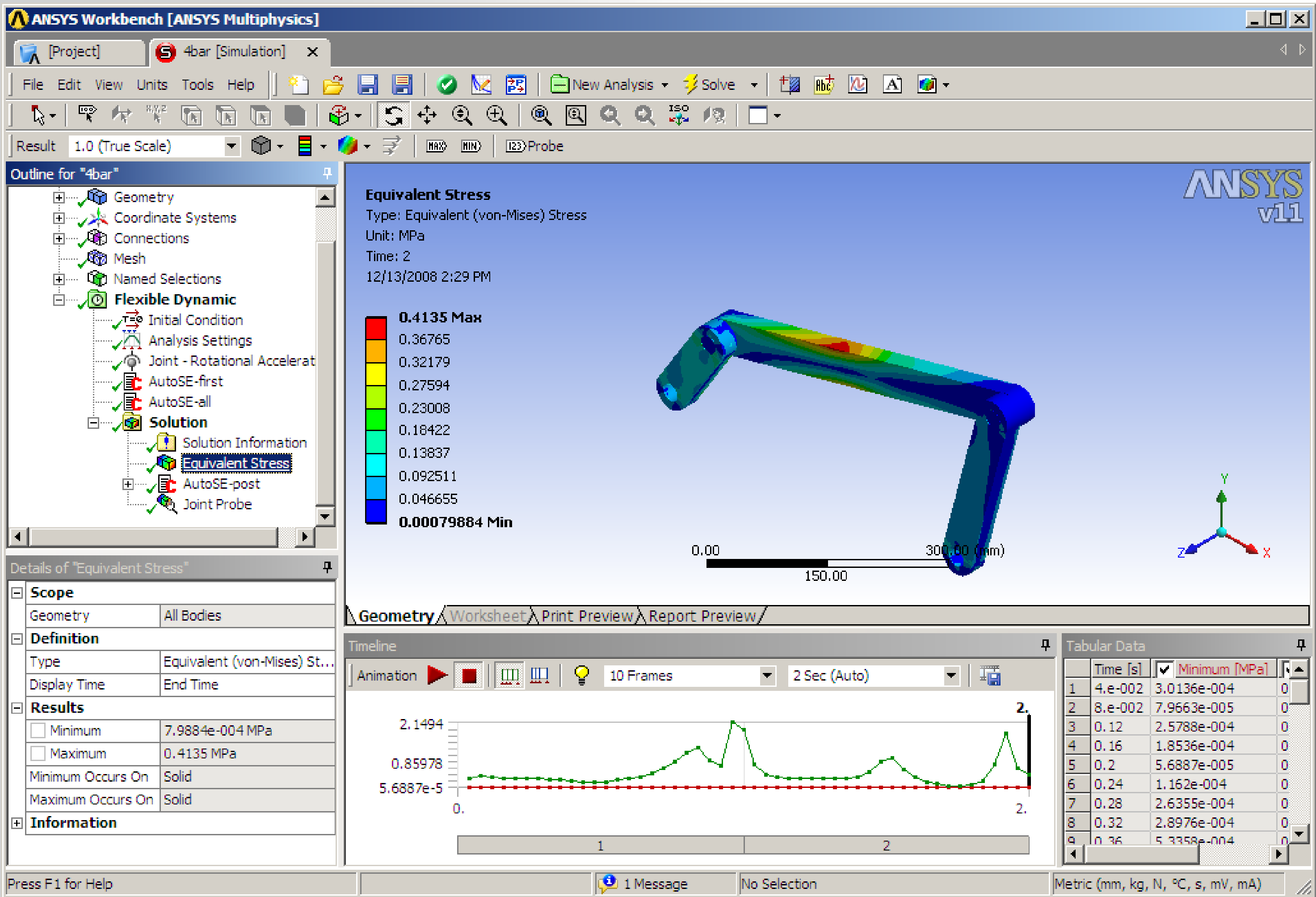
Component Mode Synthesis

- ✓ **CMS is a type of substructuring which performs a modal analysis of a structure based on independent modal analyses of its parts**

- ✓ **The synthesis involves making the components work together as a single structure by satisfying inter-component compatibility and equilibrium constraints**

- ✓ **Master DOF are required only at interface nodes**

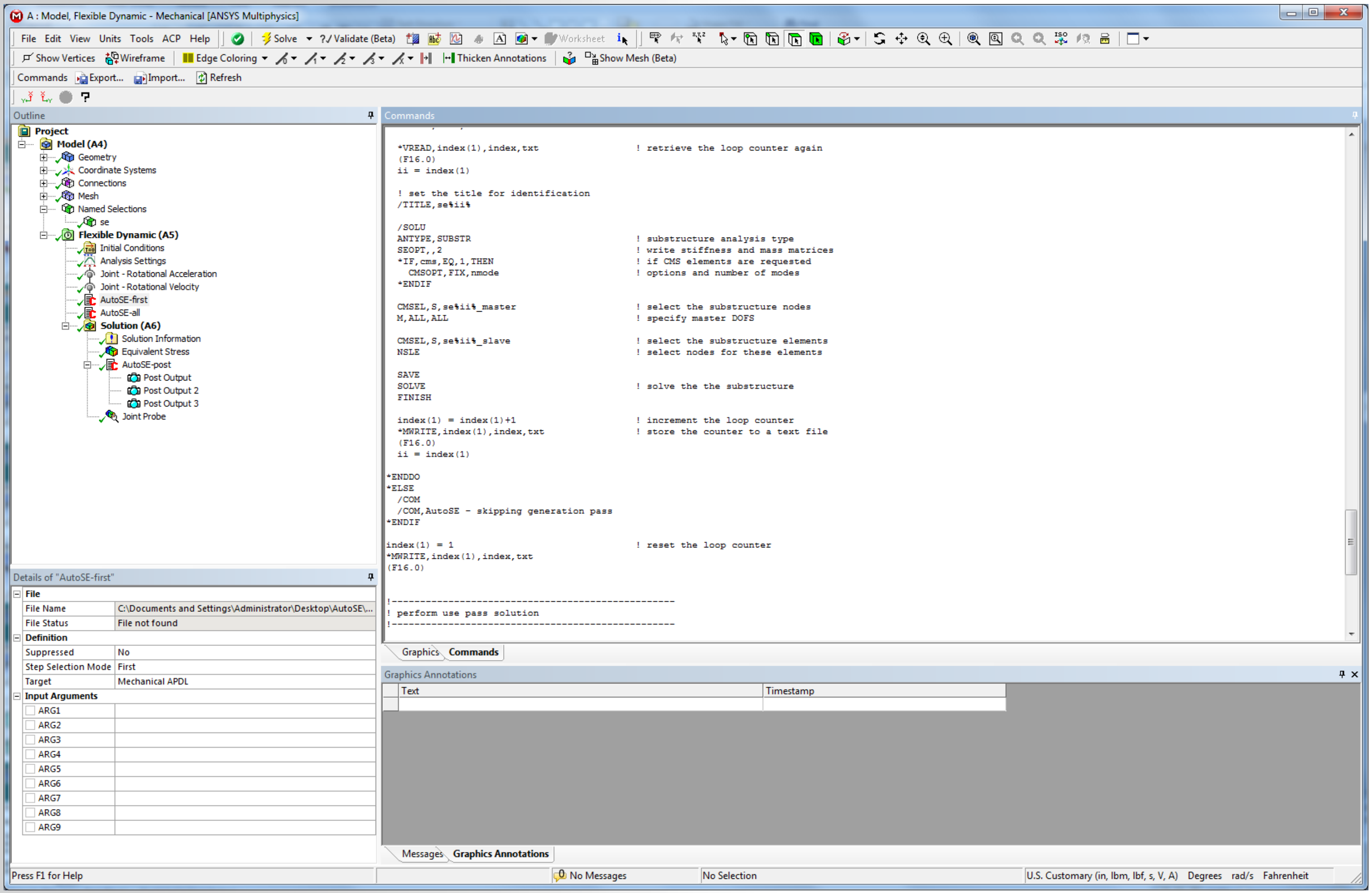
CMS with ANSYS Workbench



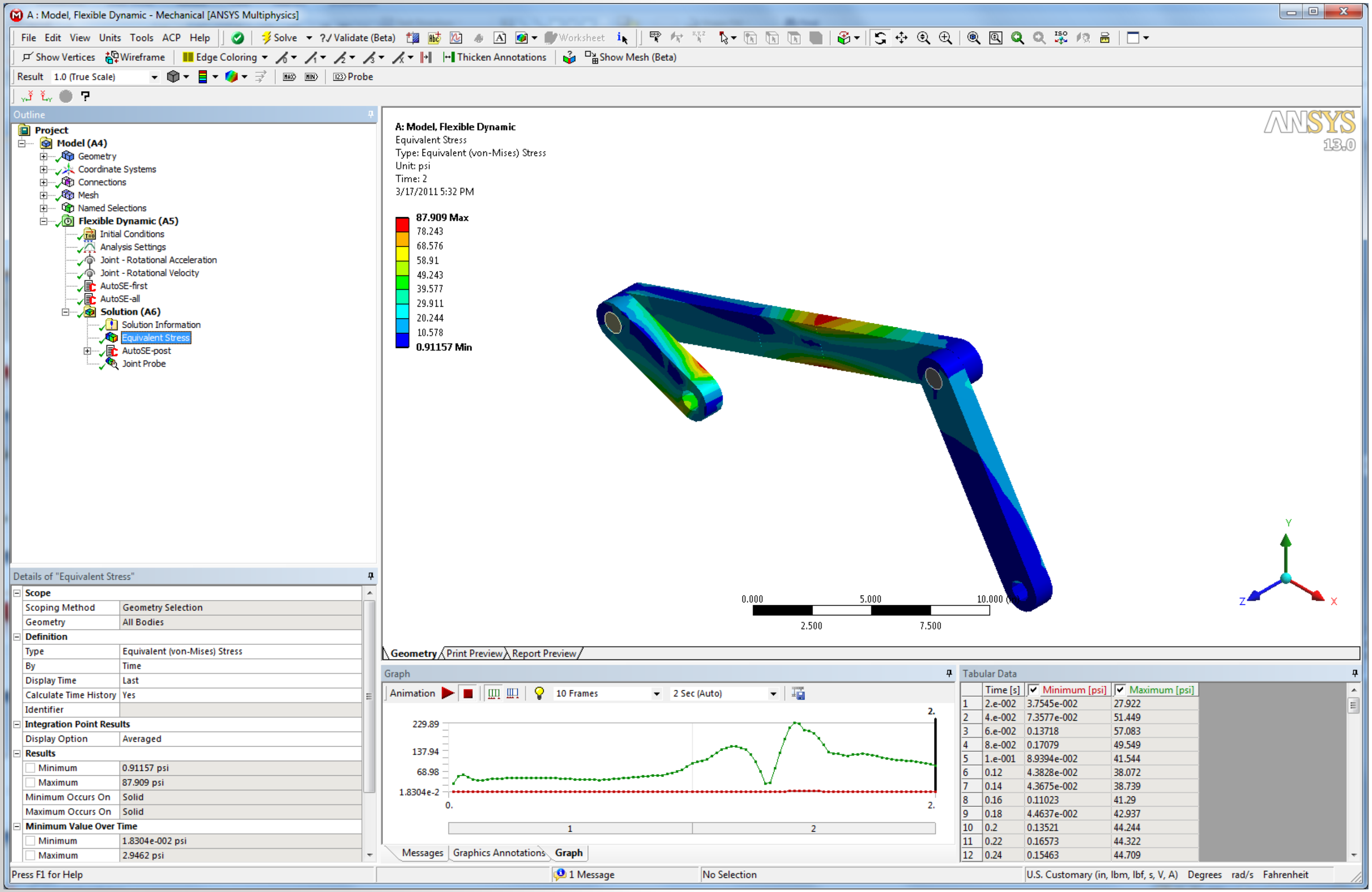
ANSYS

SVS FEM
Your partner in computing

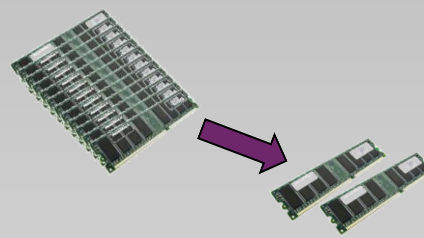
APDL macros embedded in the simulation tree for generation, use and expansion pass



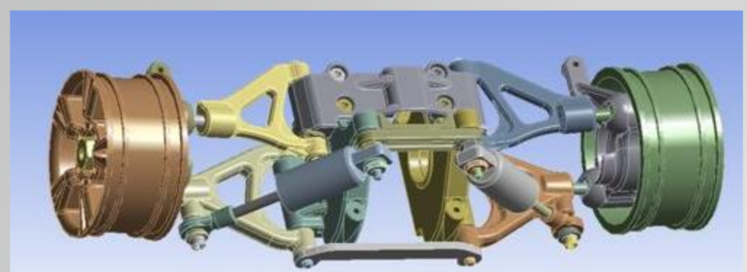
Results are available through standard operations

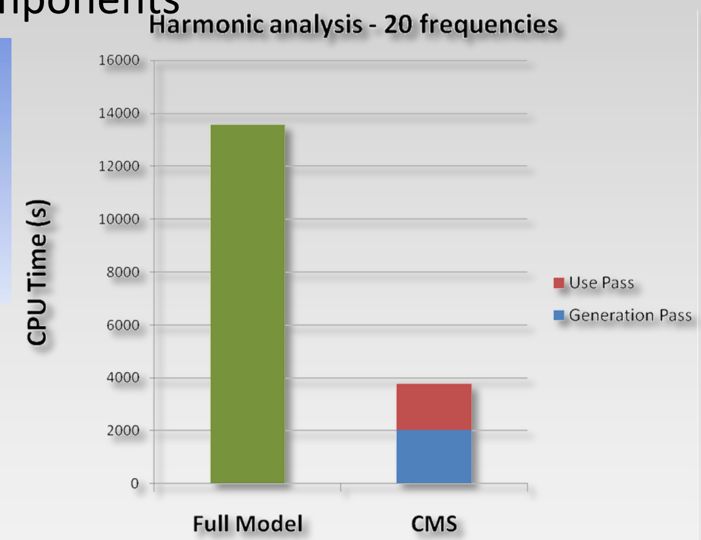


When to use substructuring?

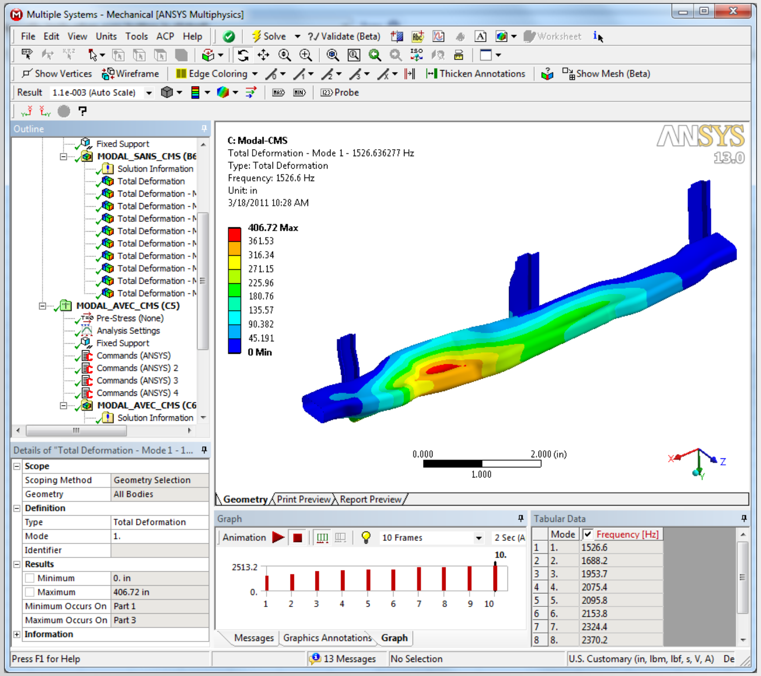


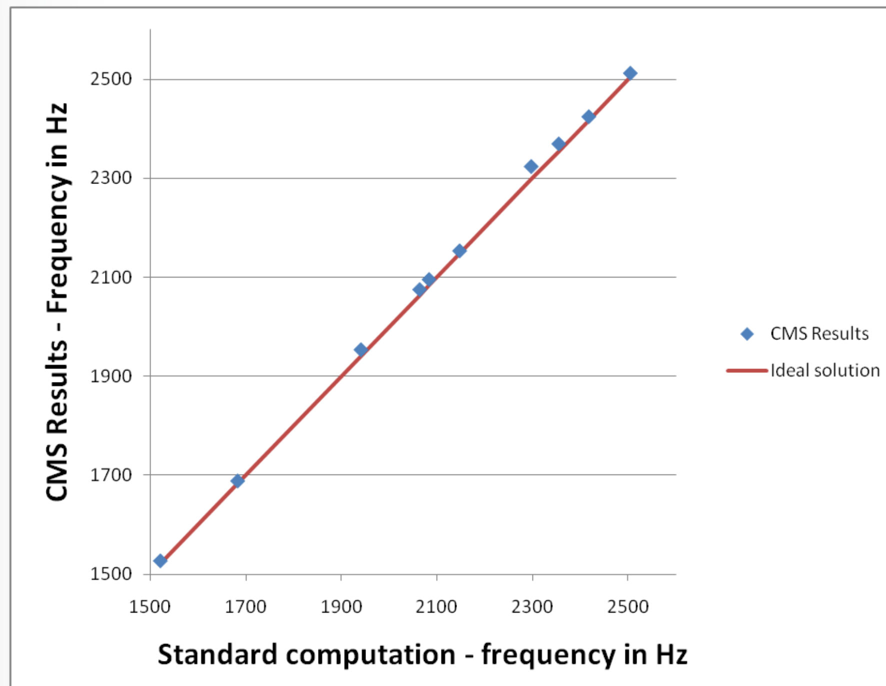
- Reduce memory consumption
- Allow collaborative work and create a library of components
- Reduce solution time for harmonic and transient analyses
- You can perform efficient design variations by reusing components





Comparing the accuracy of a CMS analysis to a standard one

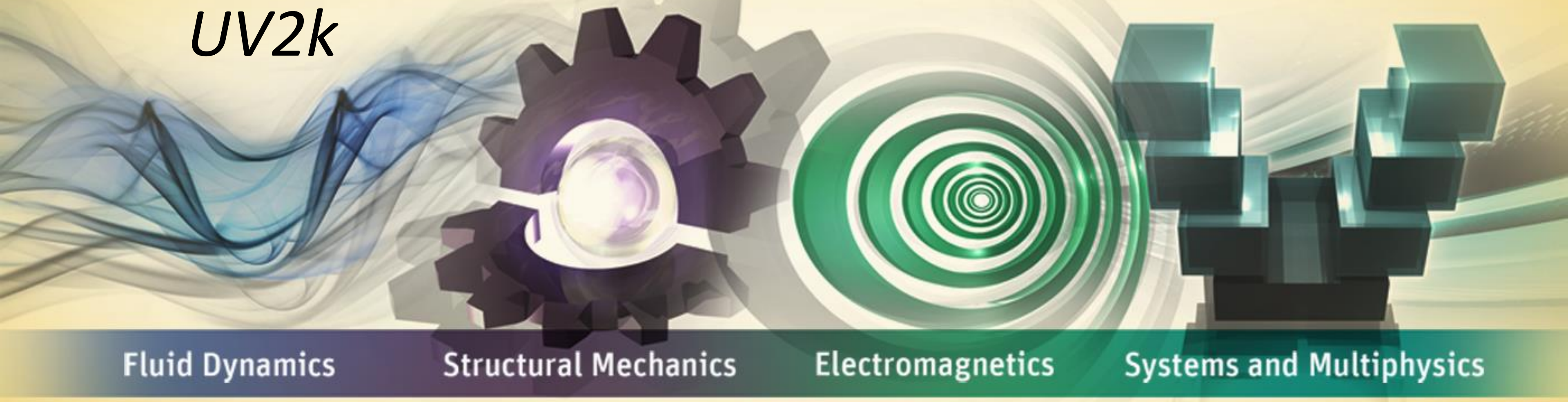




Realize Your Product Promise™

IT4Innovations

Případová studie řešení pro intenzivní HPC výpočty. Srovnání dílčích výsledků pro CERIT a UV2k



Petr Koňas

IT4Innovations&
národní
superpočítačové
centrum

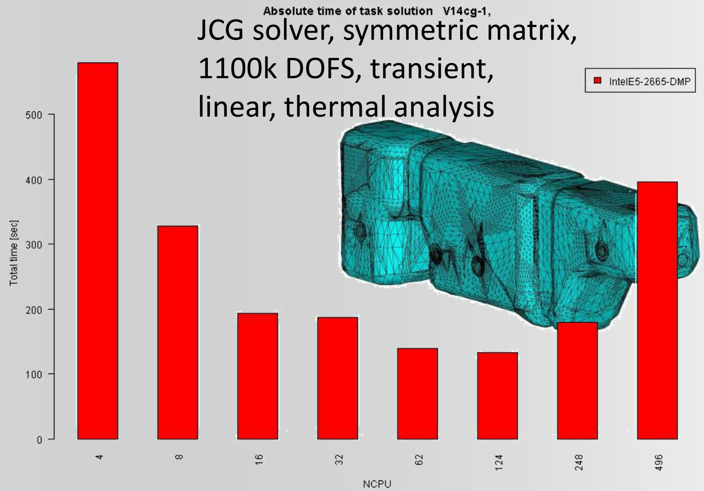
SVS FEM
Your partner in computing

Benchmarky

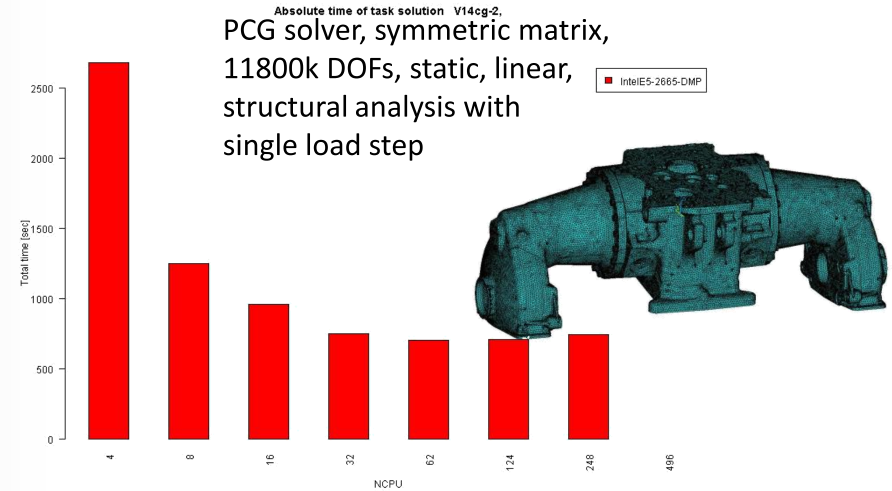
Přes 400 benchmarkových úloh. Více jak 300 stránek výsledků z benchmarků.
3 týdny testování a optimalizace.

ANSYS testy
Workbench Mechanical
CFX
Fluent
LS-DYNA

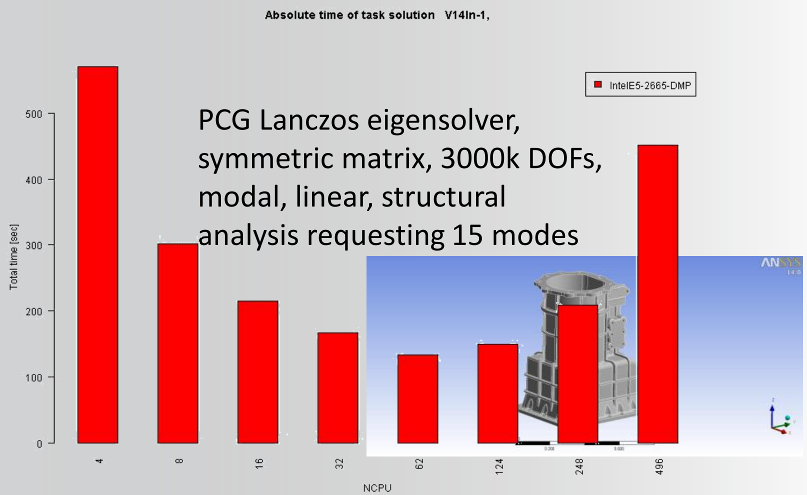
Benchmarky Workbench MAPDL



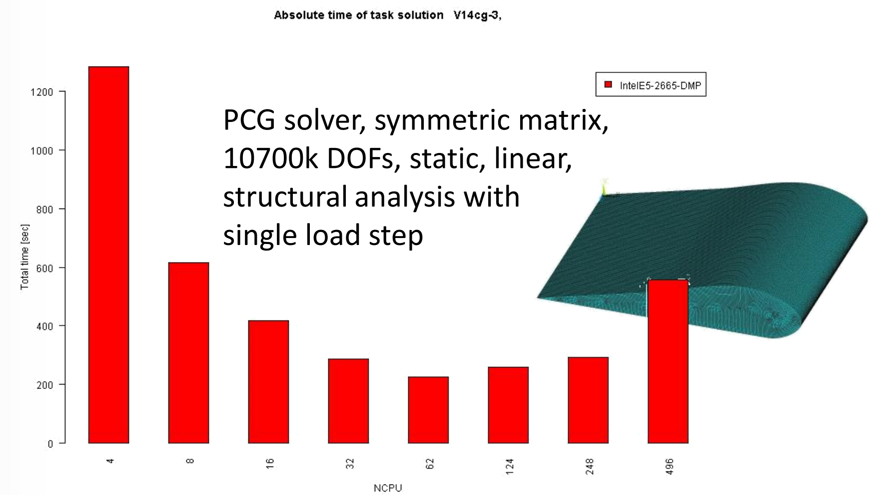
JCG solver, symmetric matrix, 1100k DOFS, transient, linear, thermal analysis



PCG solver, symmetric matrix, 11800k DOFs, static, linear, structural analysis with single load step



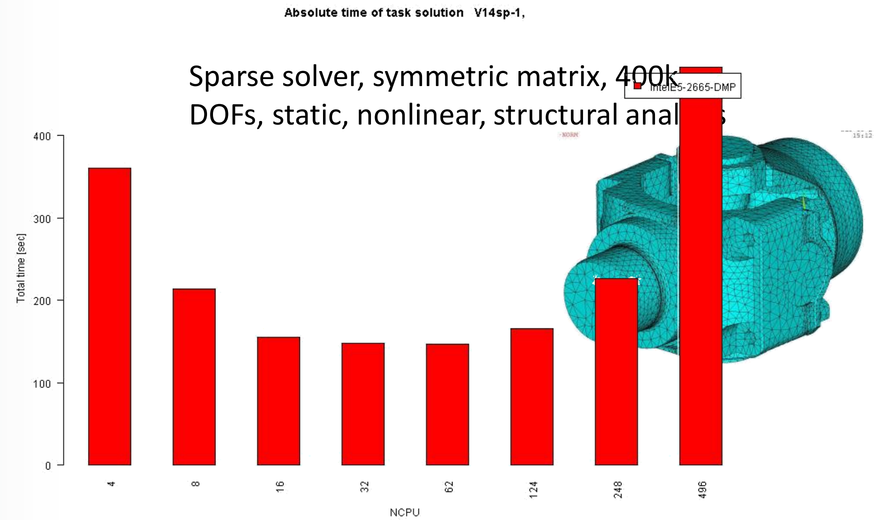
PCG Lanczos eigensolver, symmetric matrix, 3000k DOFs, modal, linear, structural analysis requesting 15 modes



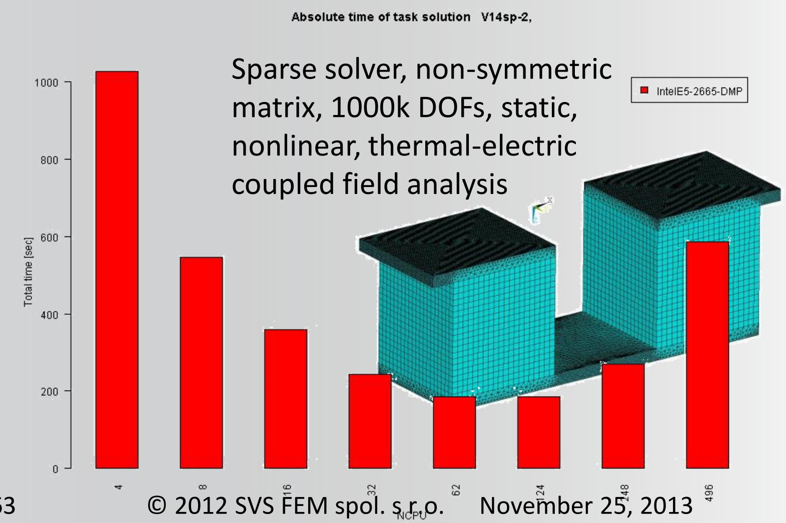
PCG solver, symmetric matrix, 10700k DOFs, static, linear, structural analysis with single load step

Benchmarky Workbench MAPDL

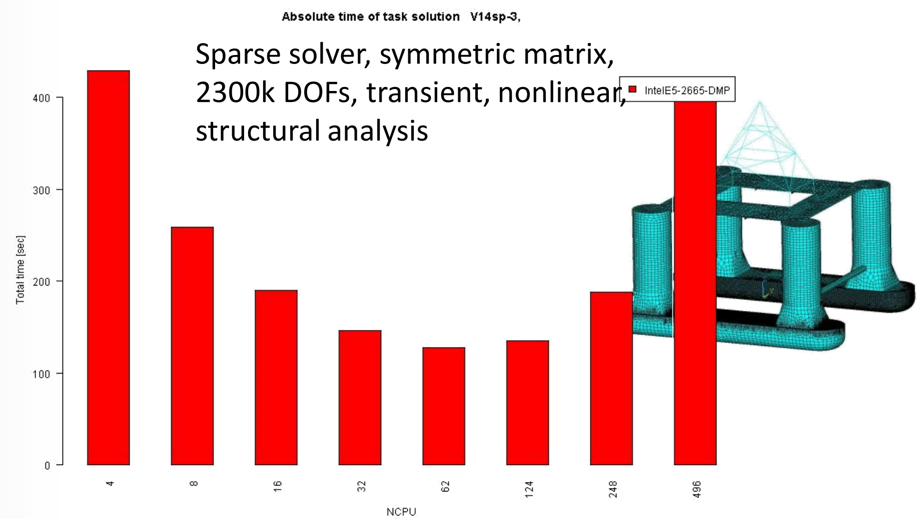
Sparse solver, symmetric matrix, 400k DOFs, static, nonlinear, structural analysis



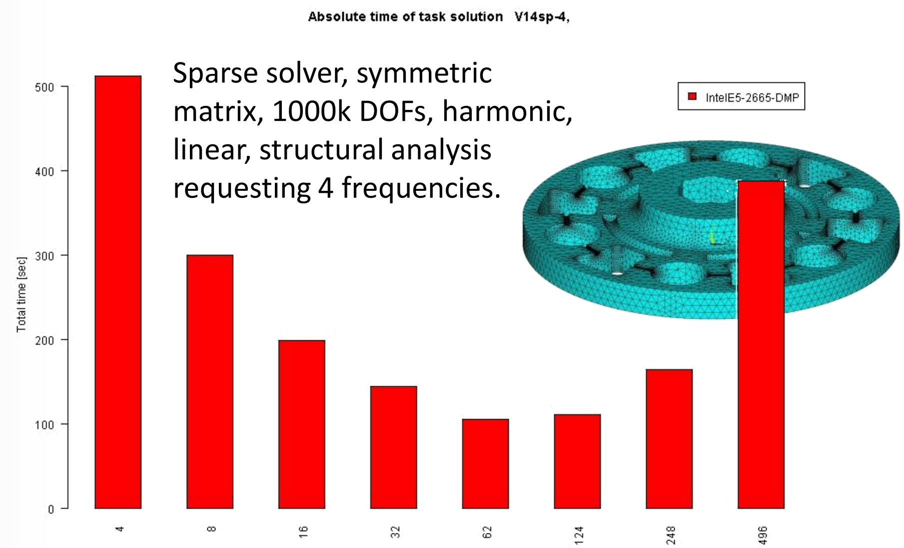
Sparse solver, non-symmetric matrix, 1000k DOFs, static, nonlinear, thermal-electric coupled field analysis



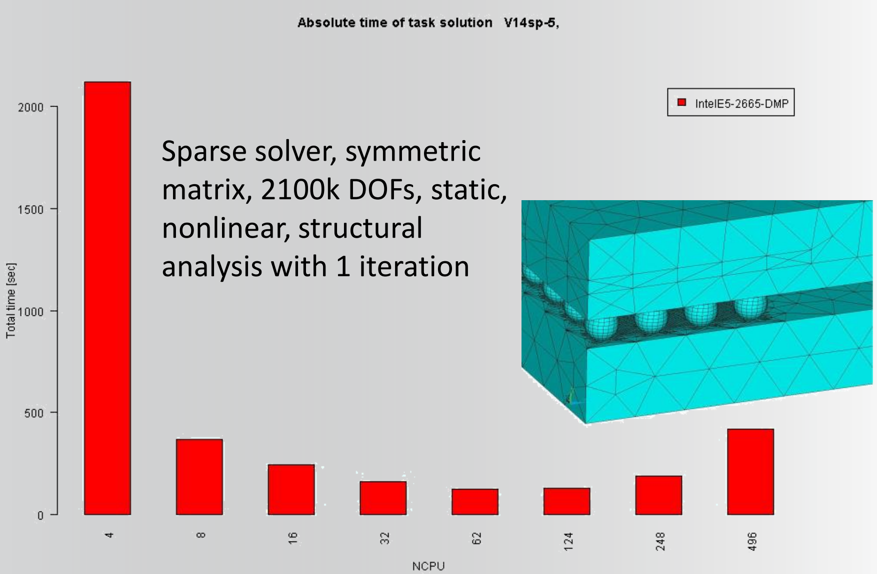
Sparse solver, symmetric matrix, 2300k DOFs, transient, nonlinear, structural analysis



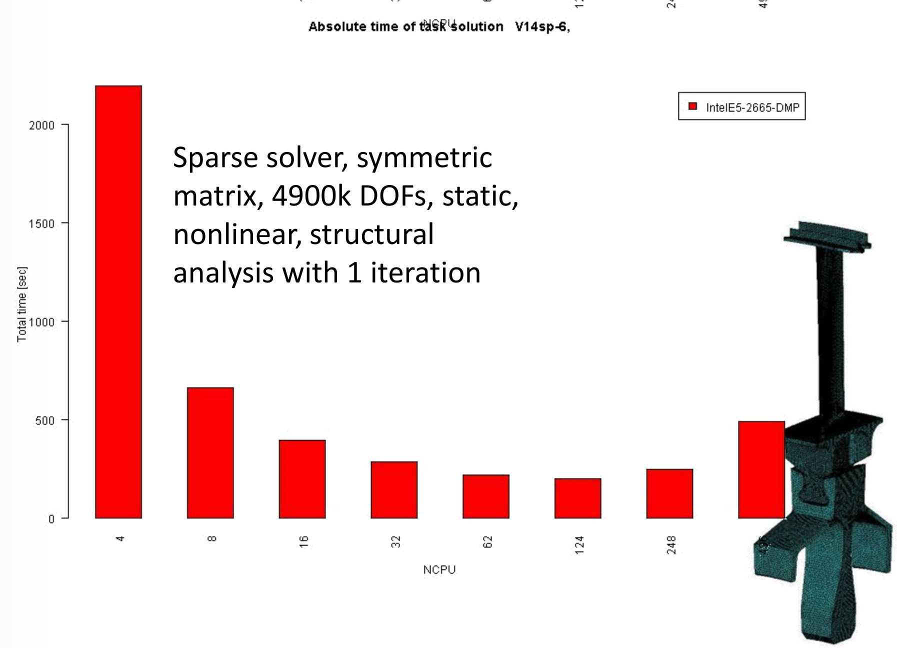
Benchmarky Workbench MAPDL



Sparse solver, symmetric matrix, 1000k DOFs, harmonic, linear, structural analysis requesting 4 frequencies.



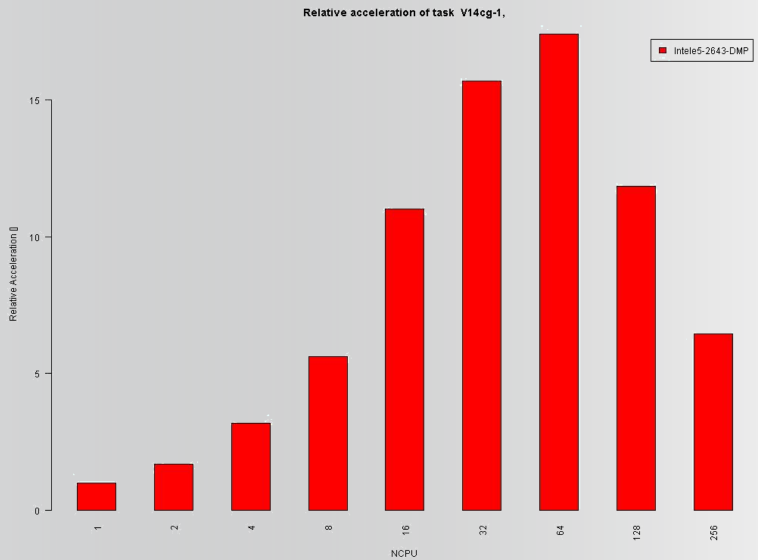
Sparse solver, symmetric matrix, 2100k DOFs, static, nonlinear, structural analysis with 1 iteration



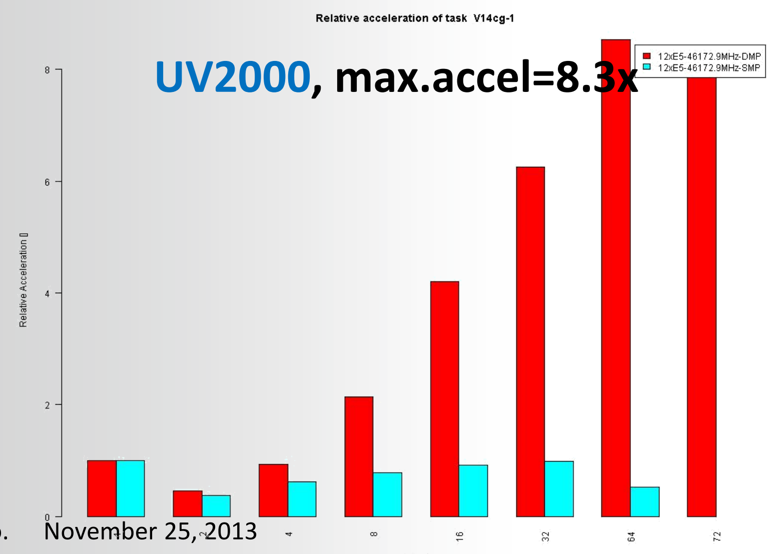
Sparse solver, symmetric matrix, 4900k DOFs, static, nonlinear, structural analysis with 1 iteration

Srovnání výsledků pro různé clustery Benchmarky Workbench MAPDL

CERIT, max.accel=17x

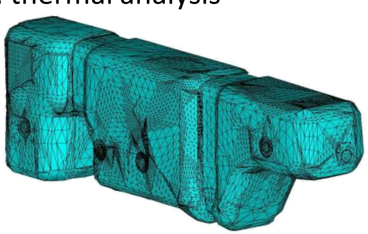


UV2000, max.accel=8.3x



CERIT=E5-2643
IT4I=E5-2665
UV2k=E5-4617

JCG solver, symmetric matrix, 1100k DOFS, transient, linear, thermal analysis

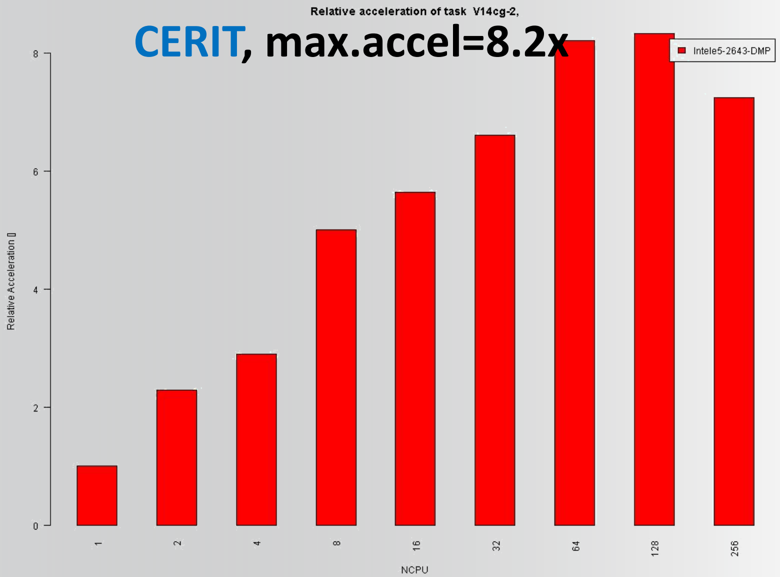


Srovnání výsledků pro různé clustery Benchmarky Workbench MAPDL

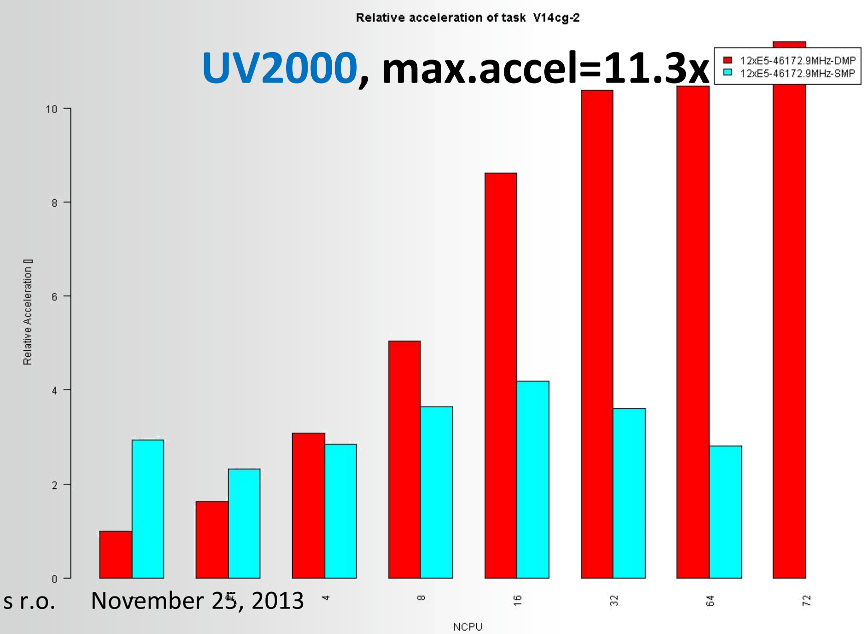
ANSYS

SVS FEM
Your partner in computing

CERIT, max.accel=8.2x

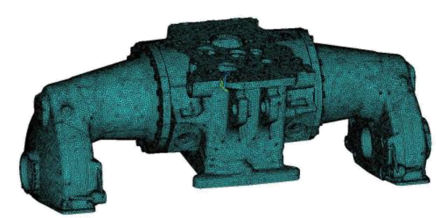


UV2000, max.accel=11.3x



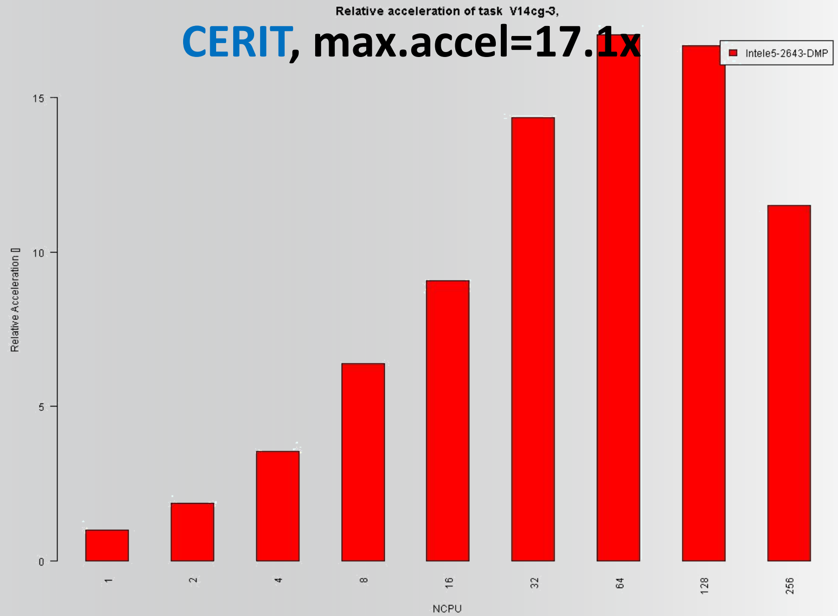
CERIT=E5-2643
IT4I=E5-2665
UV2k=E5-4617

PCG solver, symmetric matrix, 11800k DOFs, static, linear, structural analysis with single load step

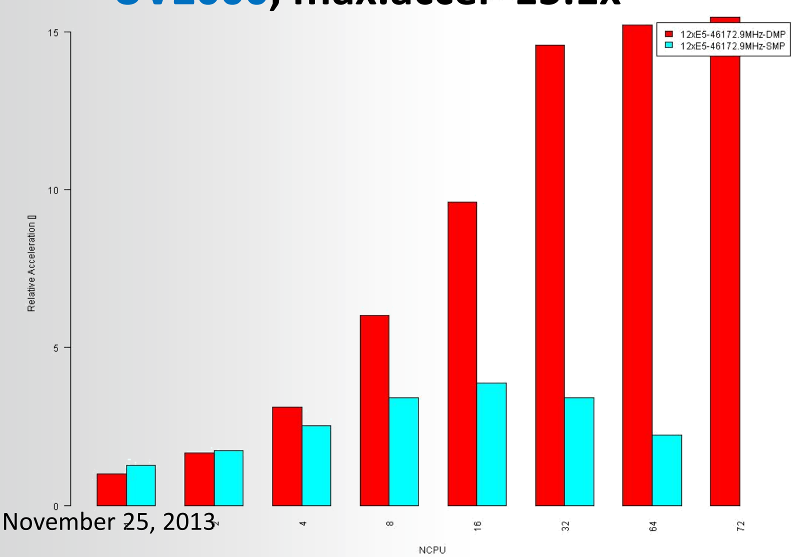


Srovnání výsledků pro různé clustery
Benchmarky Workbench MAPDL

CERIT, max.accel=17.1x



UV2000, max.accel=15.1x

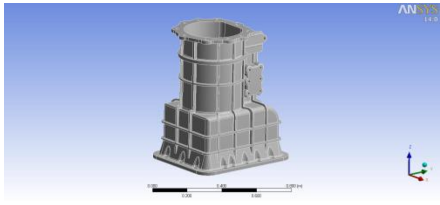


CERIT=E5-2643

IT4I=E5-2665

UV2k=E5-4617

PCG Lanczos eigensolver, symmetric matrix, 3000k DOFs, modal, linear, structural analysis requesting 15 modes

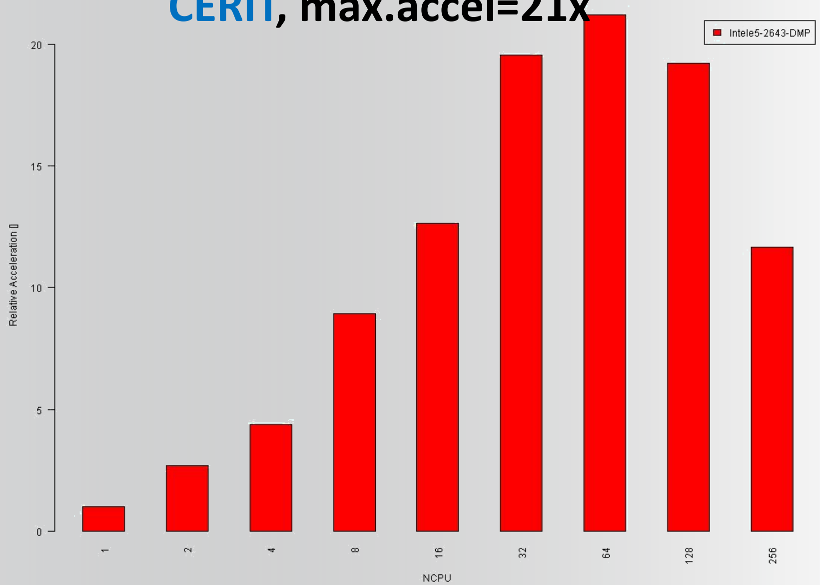


Srovnání výsledků pro různé clustery Benchmarky Workbench MAPDL

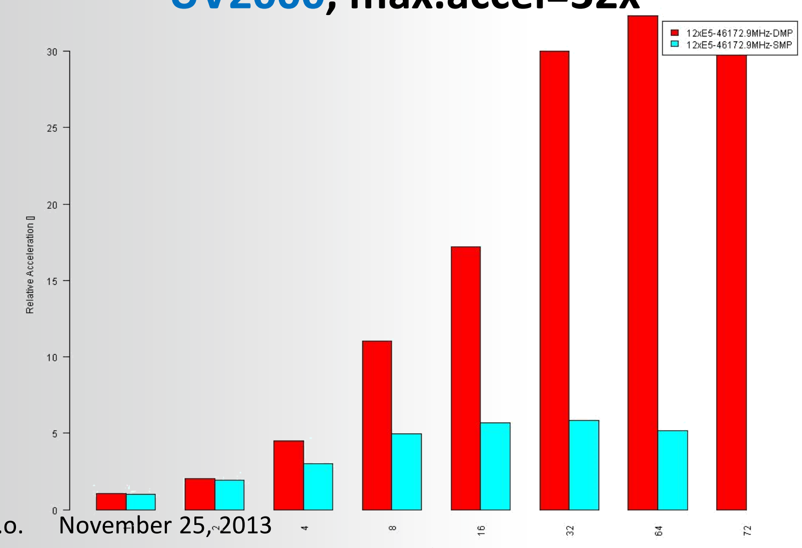
ANSYS

SVS FEM
Your partner in computing

CERIT, max.accel=21x



UV2000, max.accel=32x

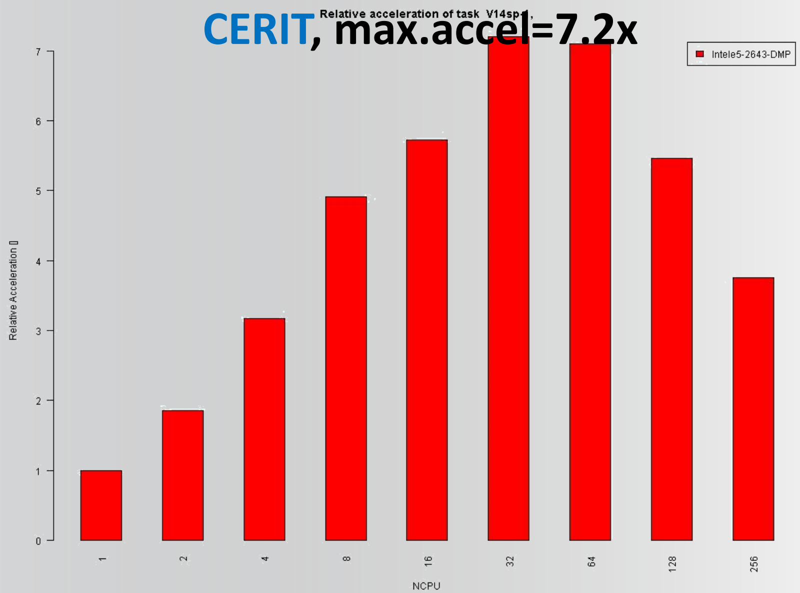


CERIT=E5-2643
IT4I=E5-2665
UV2k=E5-4617

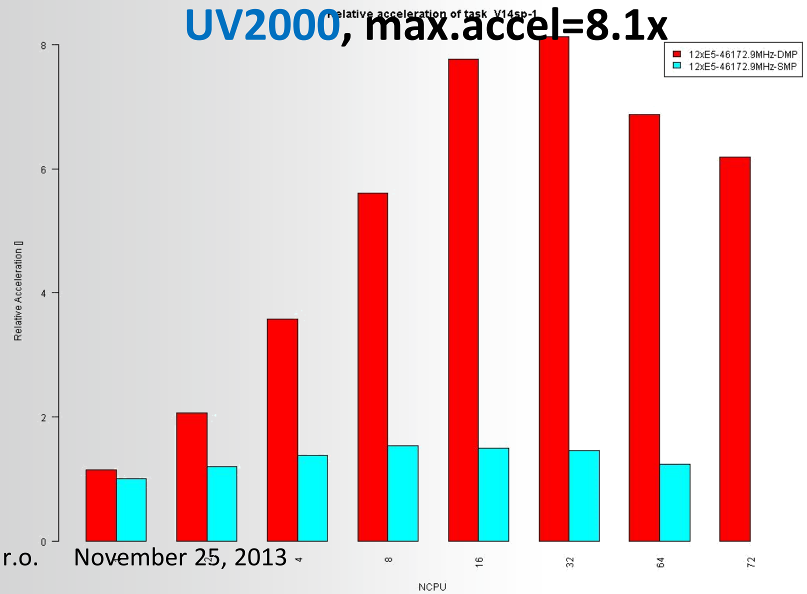
PCG solver, symmetric matrix, 10700k DOFs, static, linear, structural analysis with single load step

Srovnání výsledků pro různé clustery
Benchmarky Workbench MAPDL

CERIT, max.accel=7.2x

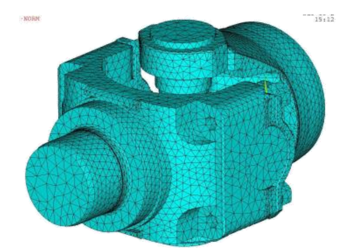


UV2000, max.accel=8.1x



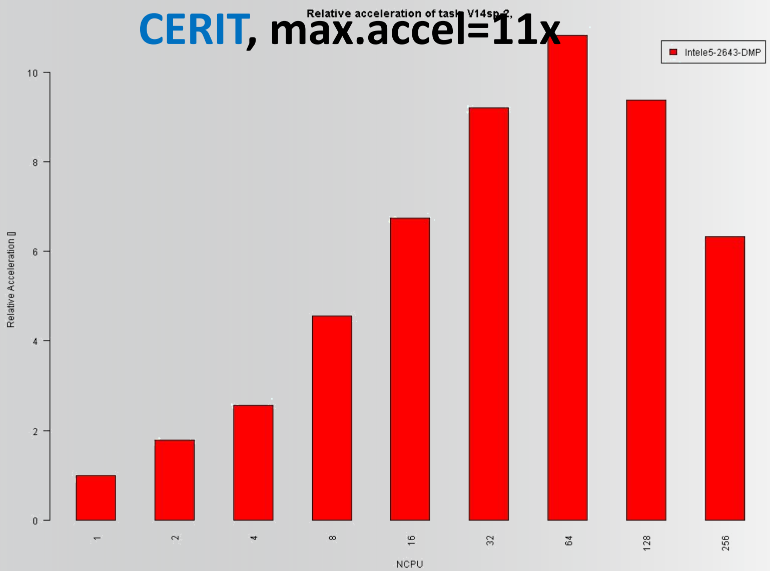
CERIT=E5-2643
IT4I=E5-2665
UV2k=E5-4617

Sparse solver, symmetric matrix, 400k DOFs, static, nonlinear, structural analysis

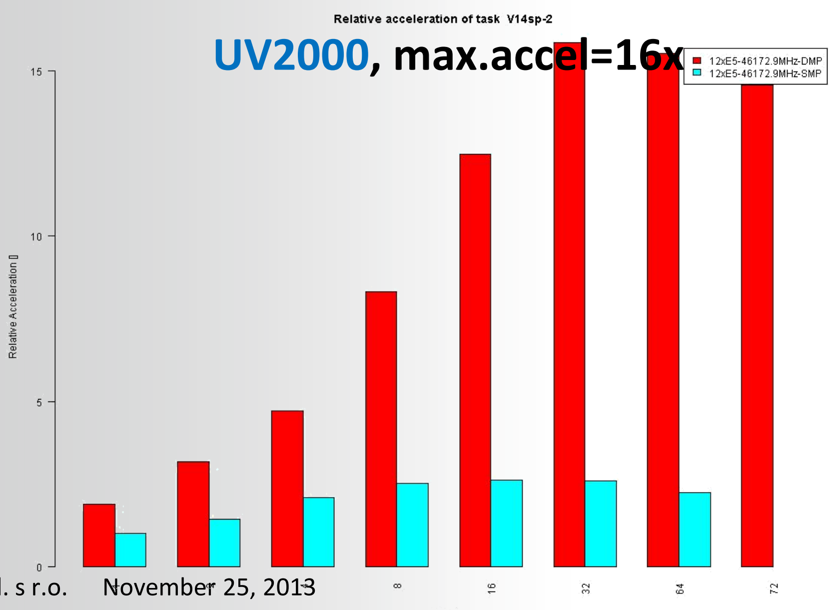


Srovnání výsledků pro různé clustery
Benchmarky Workbench MAPDL

CERIT, max.accel=11x

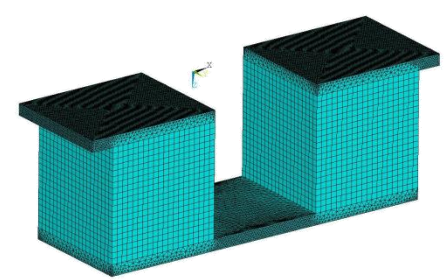


UV2000, max.accel=16x



CERIT=E5-2643
IT4I=E5-2665
UV2k=E5-4617

Sparse solver, non-symmetric matrix, 1000k DOFs, static, nonlinear, thermal-electric coupled field analysis

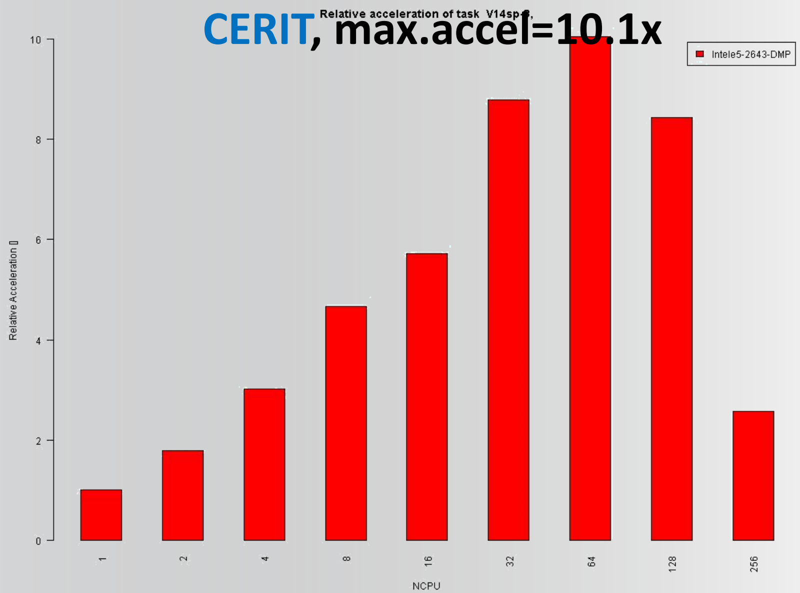


Srovnání výsledků pro různé clustery Benchmarky Workbench MAPDL

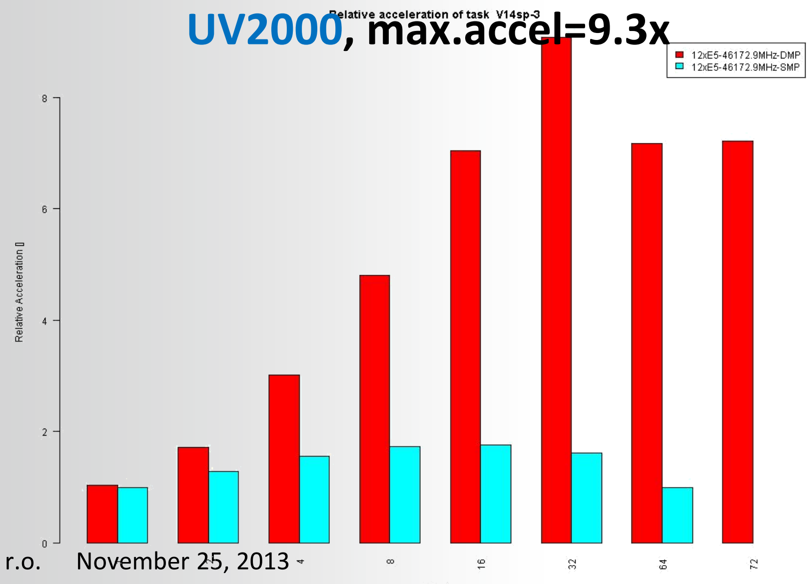
ANSYS

SVS FEM
Your partner in computing

CERIT, max.accel=10.1x

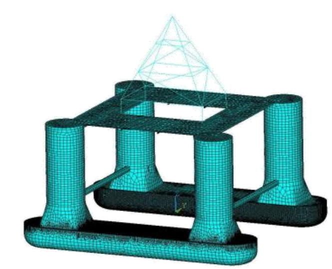


UV2000, max.accel=9.3x



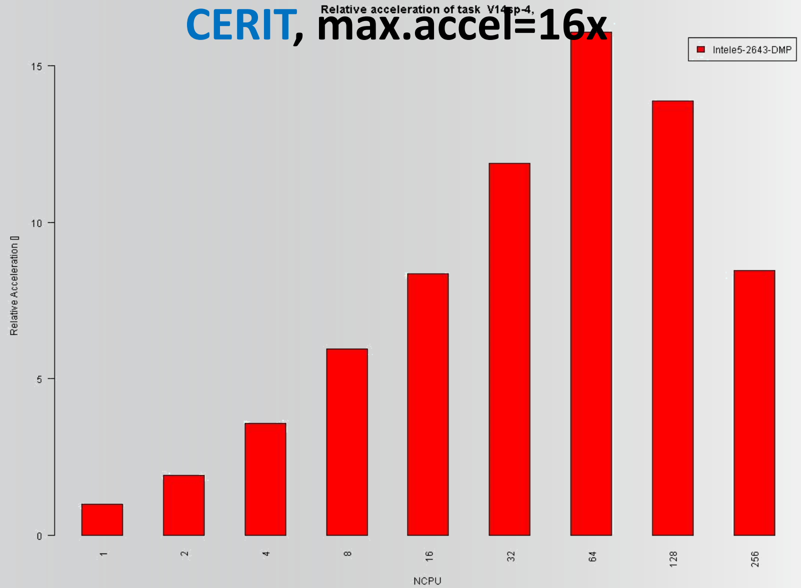
CERIT=E5-2643
IT4I=E5-2665
UV2k=E5-4617

Sparse solver, symmetric matrix, 2300k DOFs, transient, nonlinear, structural analysis

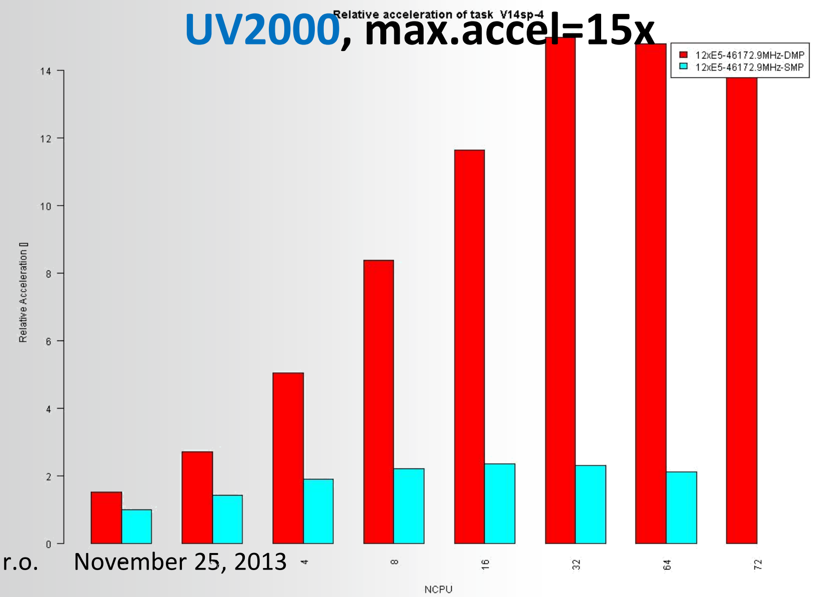


Srovnání výsledků pro různé clustery
Benchmarky Workbench MAPDL

CERIT, max.accel=16x

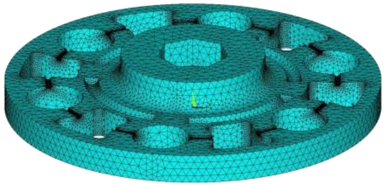


UV2000, max.accel=15x



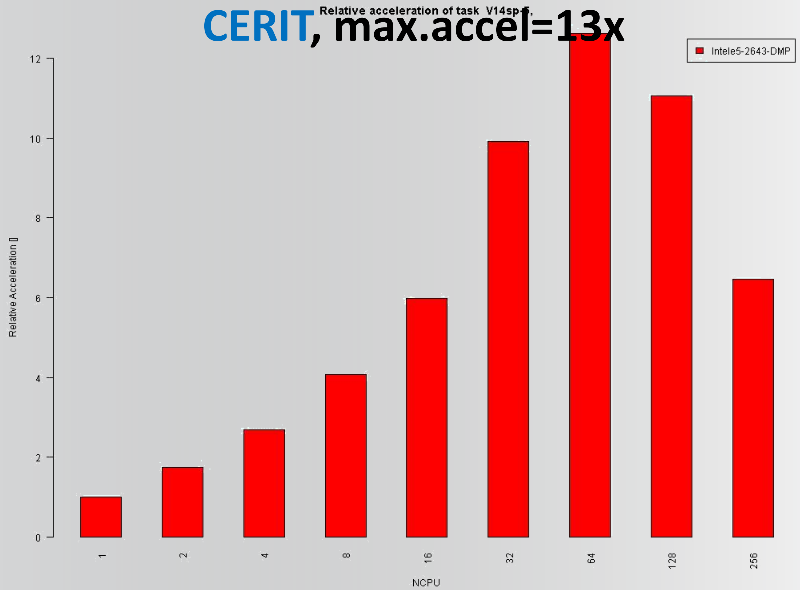
CERIT=E5-2643
IT4I=E5-2665
UV2k=E5-4617

Sparse solver, symmetric matrix, 1000k DOFs, harmonic, linear, structural analysis requesting 4 frequencies.

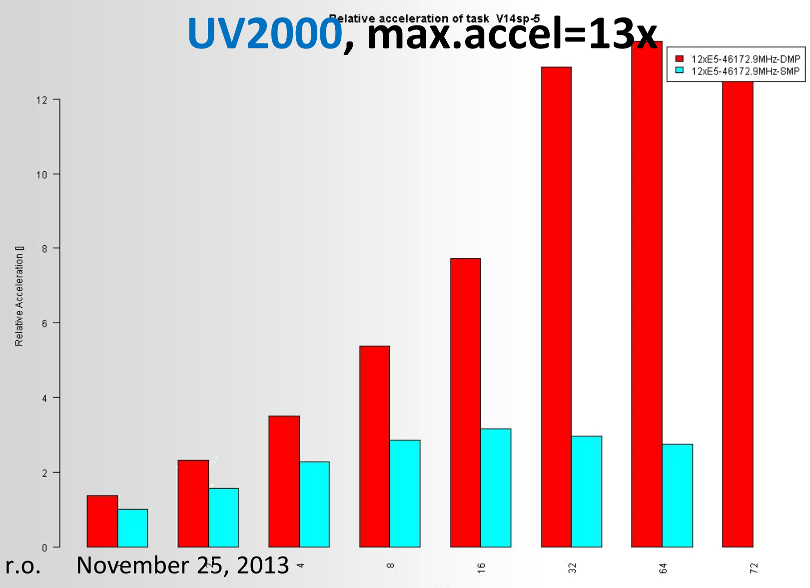


Srovnání výsledků pro různé clustery Benchmarky Workbench MAPDL

CERIT, max.accel=13x

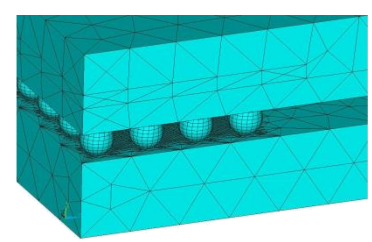


UV2000, max.accel=13x



CERIT=E5-2643
IT4I=E5-2665
UV2k=E5-4617

Sparse solver, symmetric matrix, 2100k DOFs, static, nonlinear, structural analysis with 1 iteration

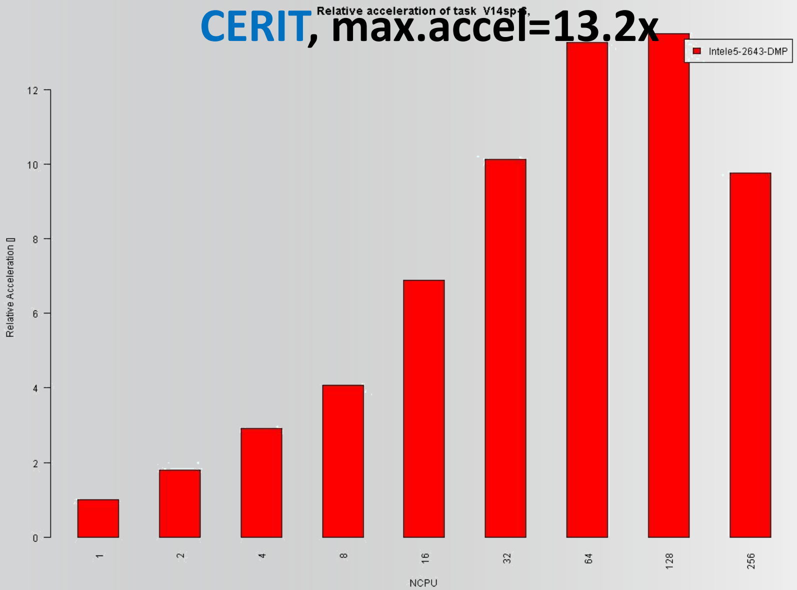


Srovnání výsledků pro různé clustery
Benchmarky Workbench MAPDL

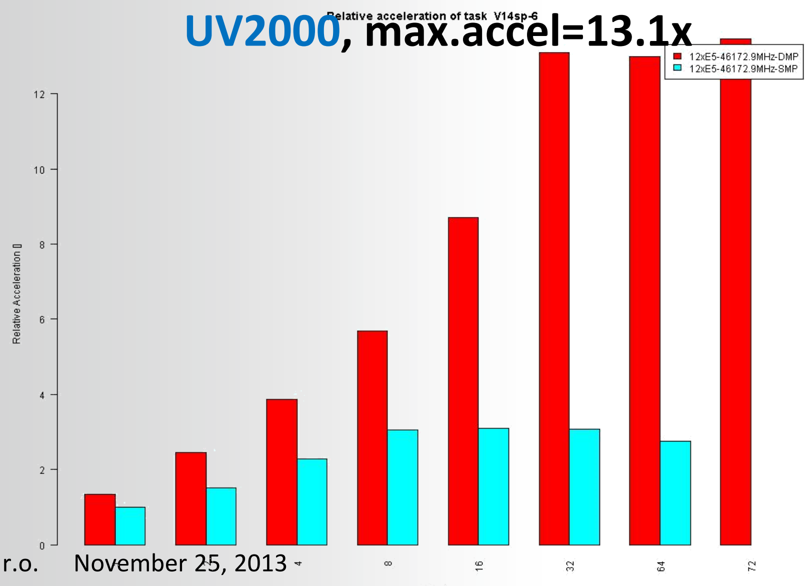
ANSYS

SVS FEM
Your partner in computing

CERIT, max.accel=13.2x



UV2000, max.accel=13.1x

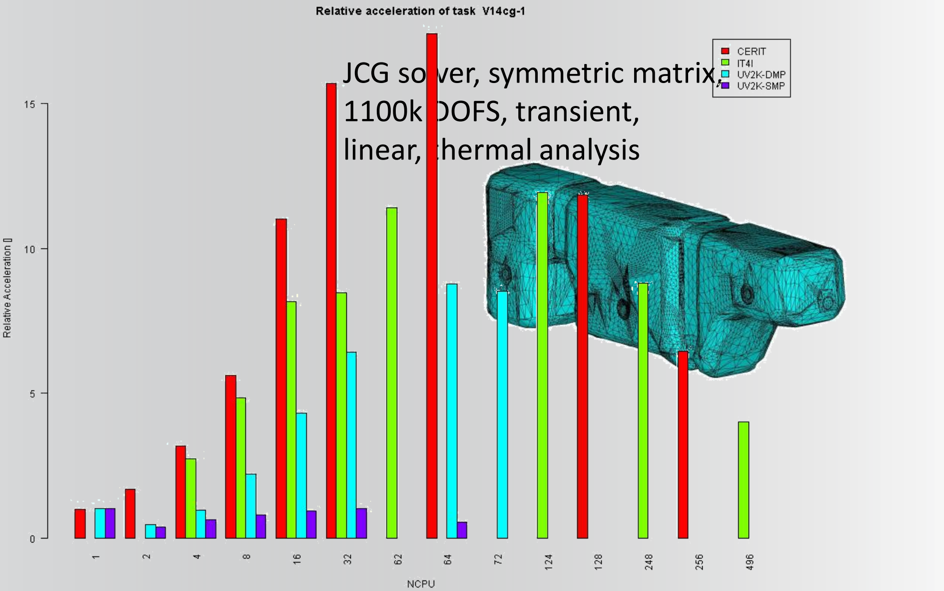


CERIT=E5-2643
IT4I=E5-2665
UV2k=E5-4617

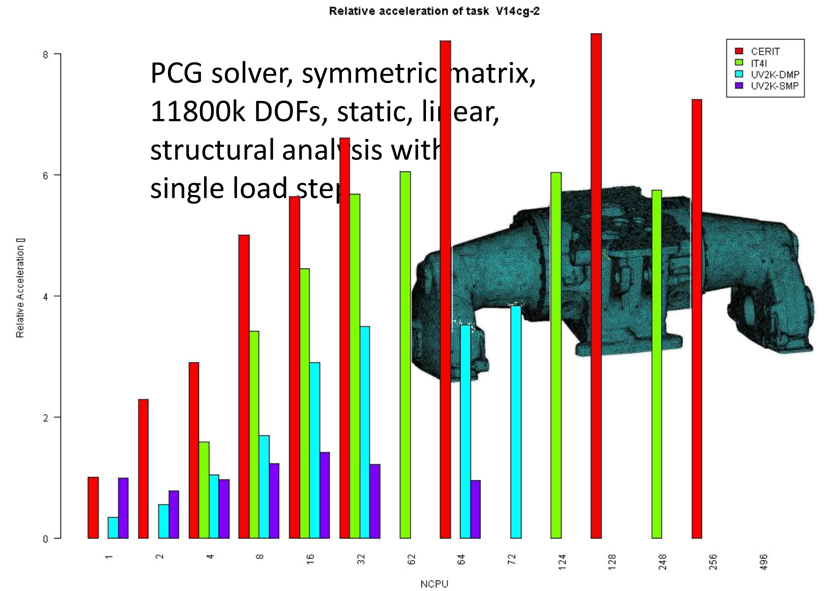
Sparse solver, symmetric matrix, 4900k DOFs, static, nonlinear, structural analysis with 1 iteration

Srovnání výsledků pro různé clustery
Benchmarky Workbench MAPDL, normalizované výsledky

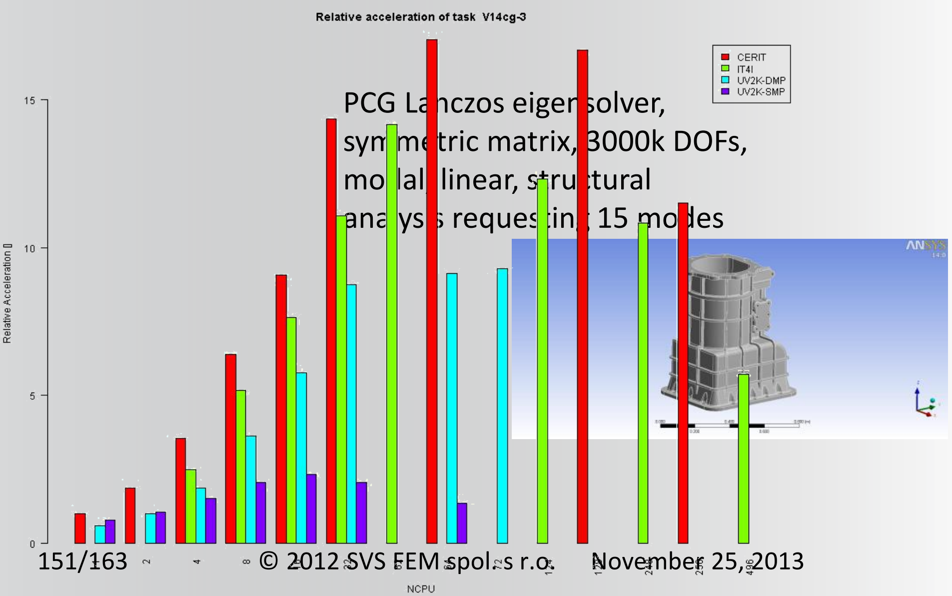
JCG solver, symmetric matrix, 1100k DOFS, transient, linear, thermal analysis



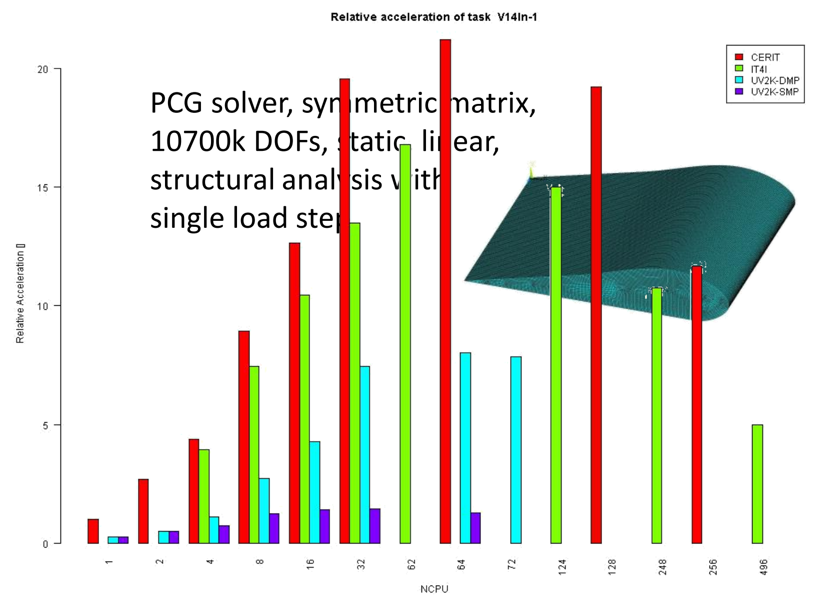
PCG solver, symmetric matrix, 11800k DOFs, static, linear, structural analysis with single load step



PCG Lanczos eigensolver, symmetric matrix, 3000k DOFs, modal, linear, structural analysis requesting 15 modes

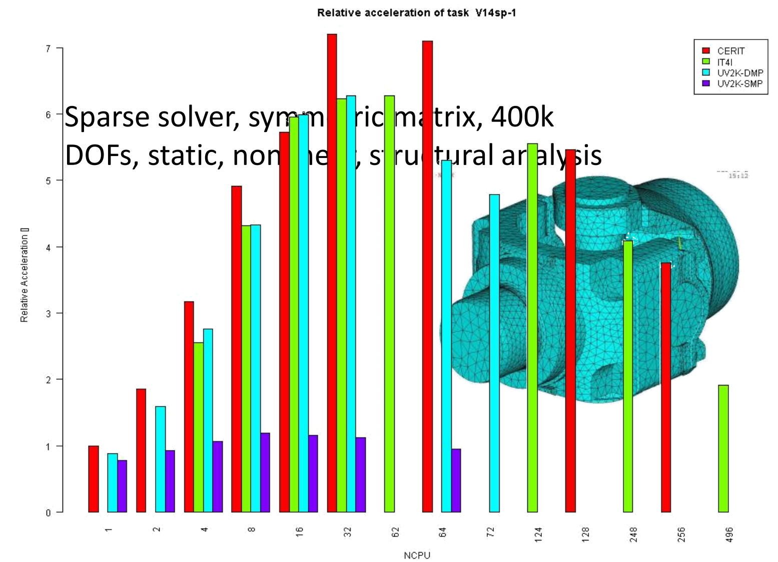


PCG solver, symmetric matrix, 10700k DOFs, static, linear, structural analysis with single load step

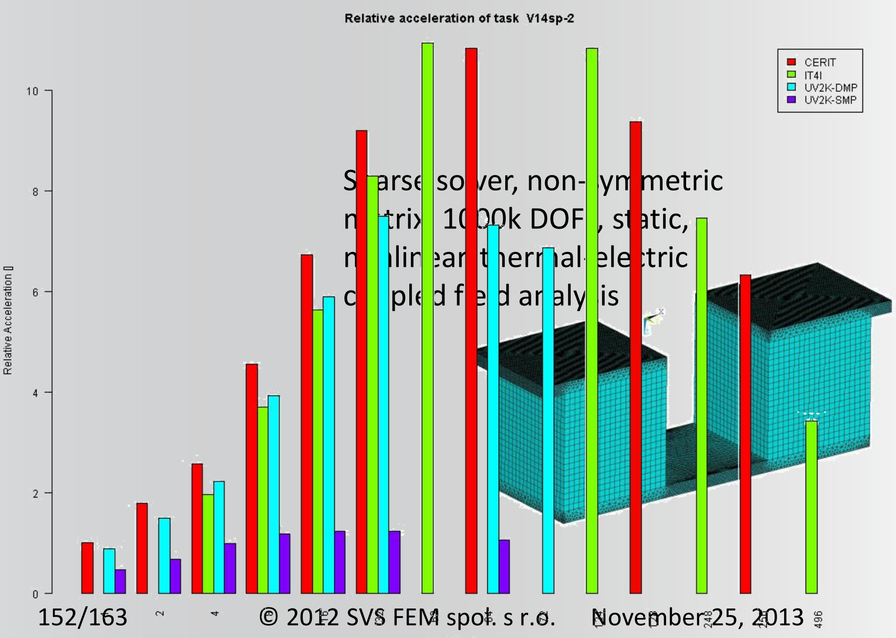


CERIT=E5-2643, **IT4I**=E5-2665,**UV2k**=E5-4617

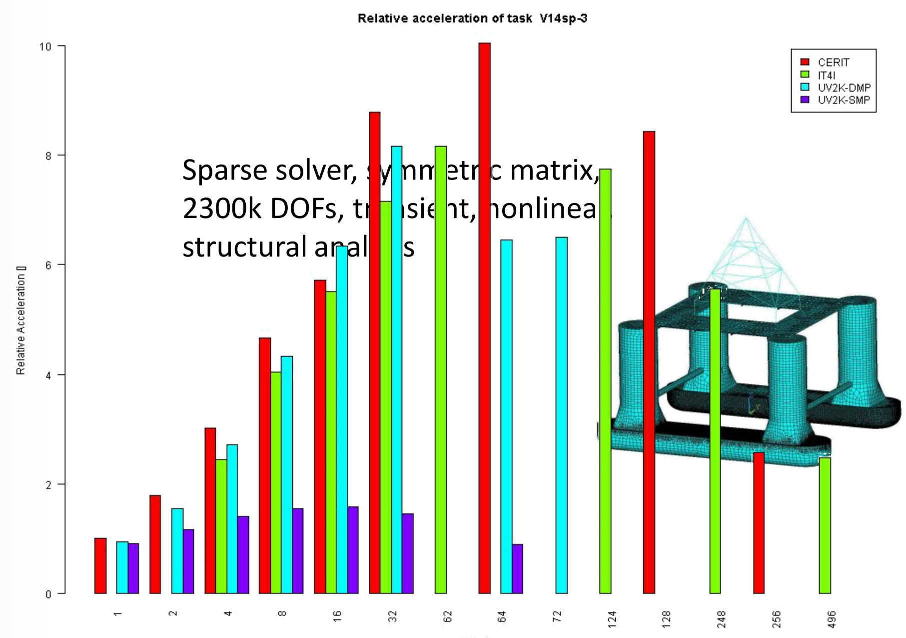
Srovnání výsledků pro různé clustery
Benchmarky Workbench MAPDL, normalizované výsledky



Sparse solver, symmetric matrix, 400k DOFs, static, nonlinear, structural analysis

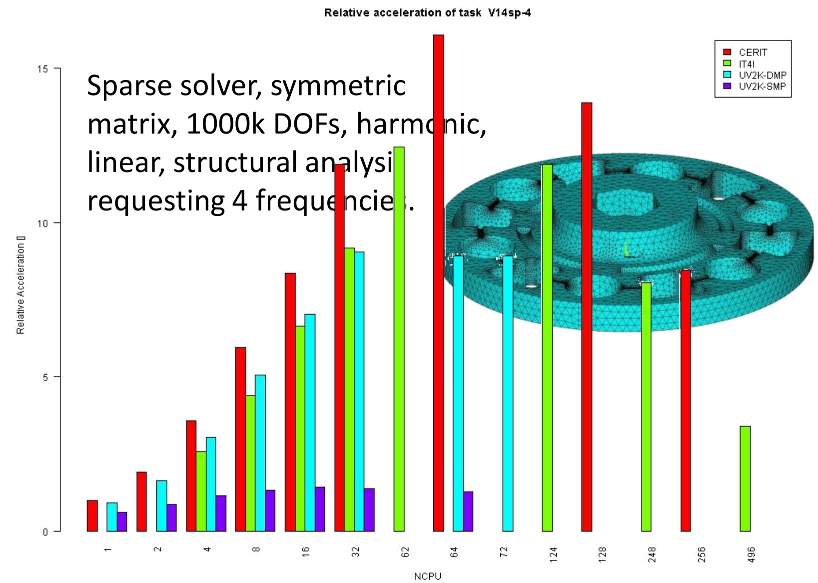


Sparse solver, non-symmetric matrix, 1000k DOFs, static, nonlinear thermal-electric coupled field analysis

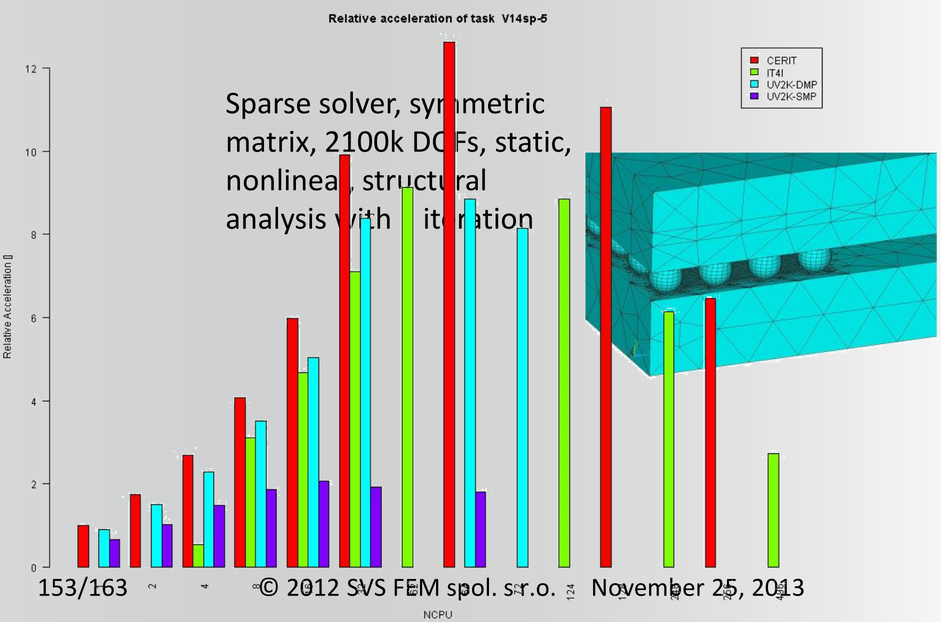


Sparse solver, symmetric matrix, 2300k DOFs, transient, nonlinear structural analysis

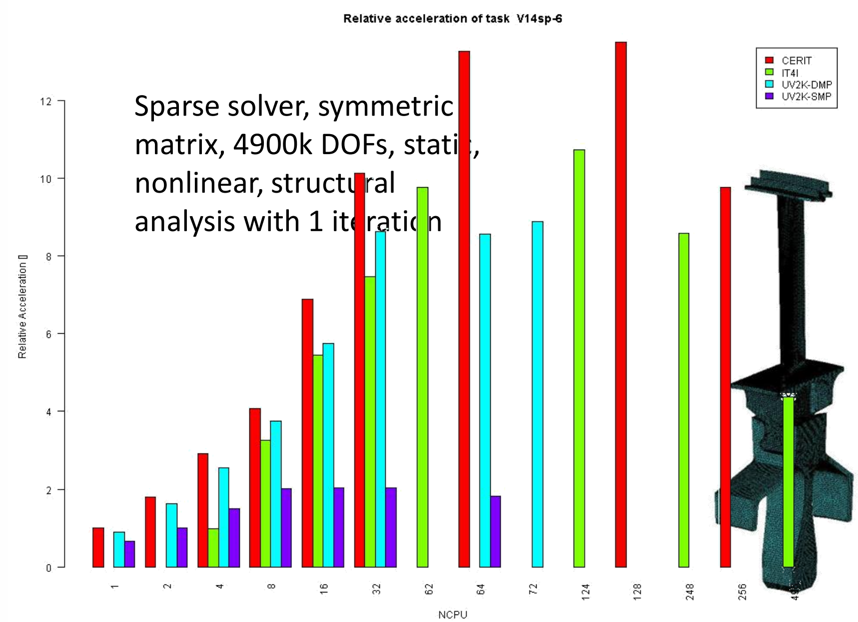
Srovnání výsledků pro různé clustery
Benchmarky Workbench MAPDL, normalizované výsledky



Sparse solver, symmetric matrix, 1000k DOFs, harmonic, linear, structural analysis requesting 4 frequencies.

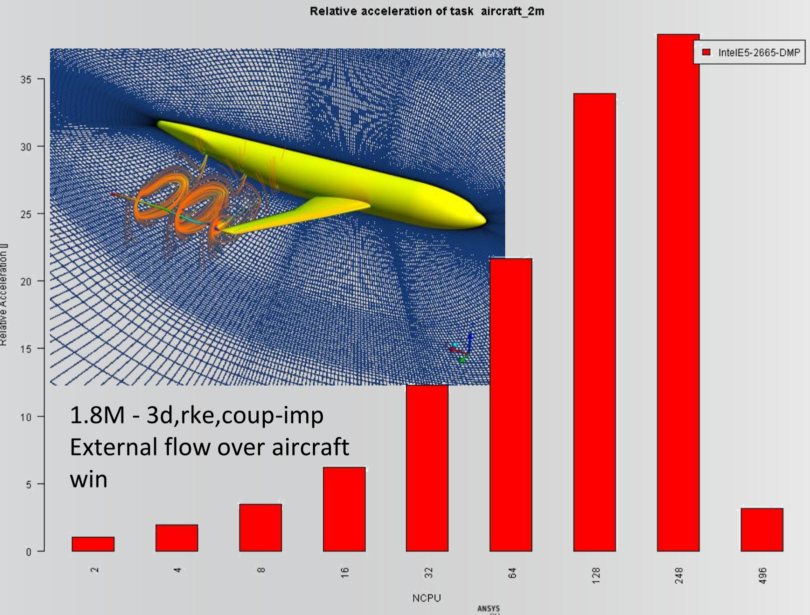


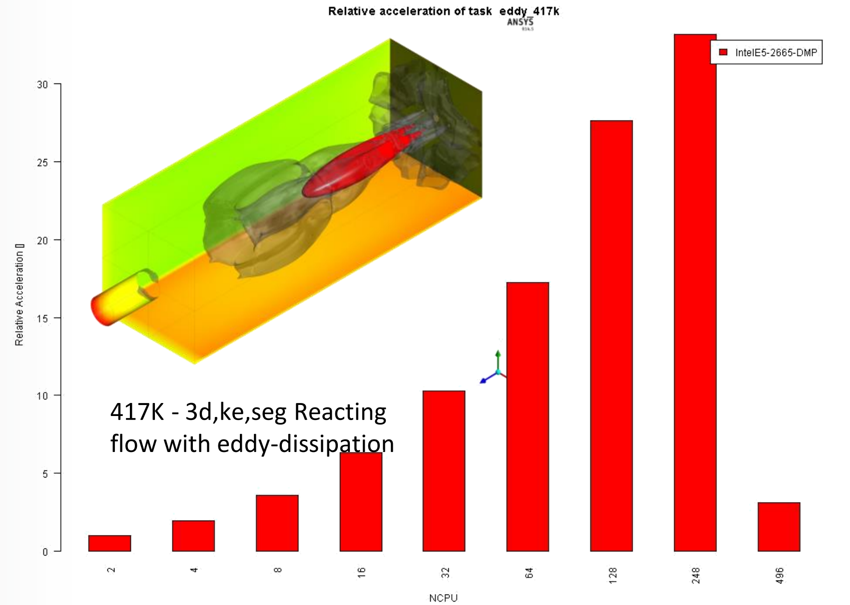
Sparse solver, symmetric matrix, 2100k DOFs, static, nonlinear, structural analysis with 1 iteration

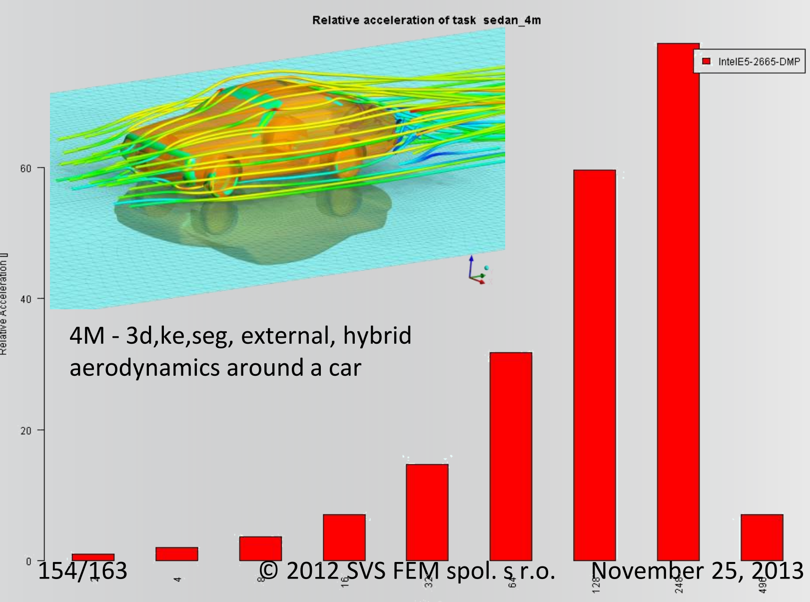


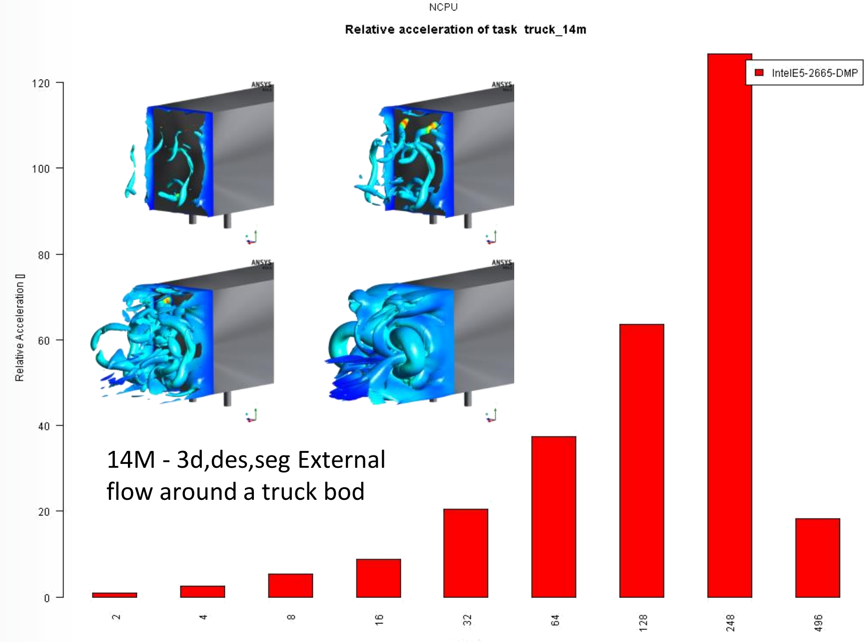
Sparse solver, symmetric matrix, 4900k DOFs, static, nonlinear, structural analysis with 1 iteration

Benchmarky Fluent

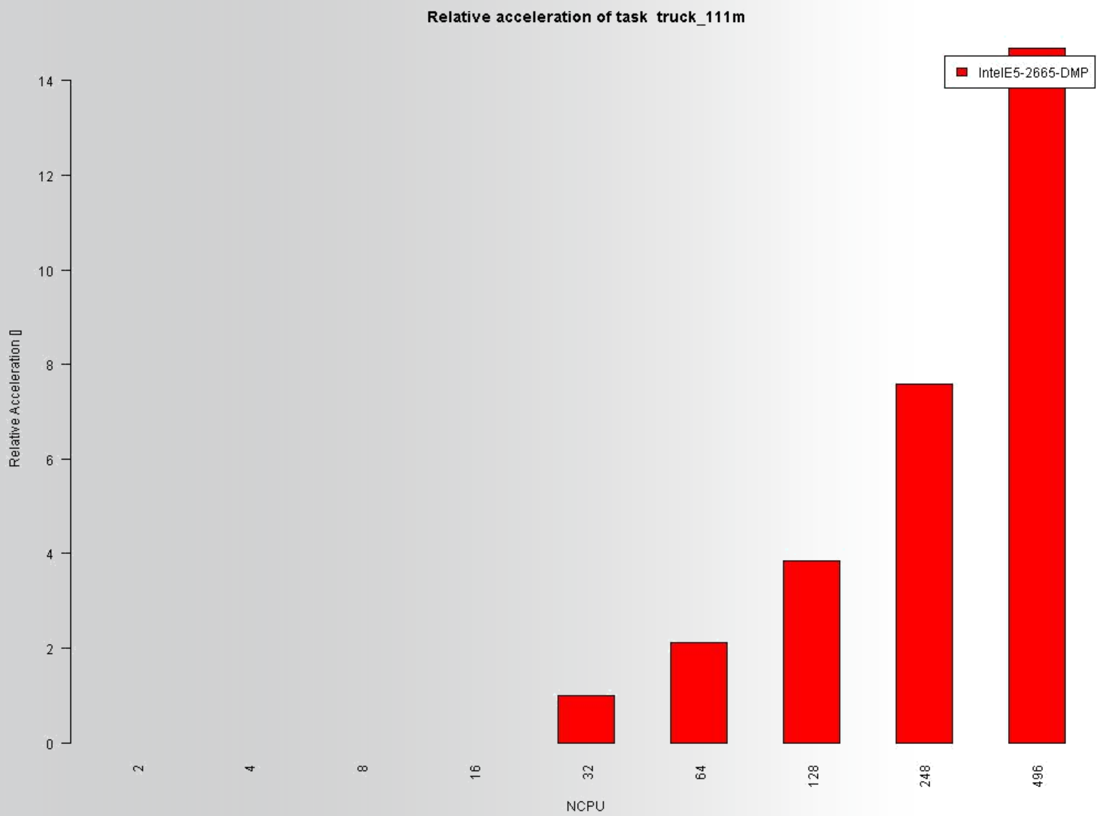




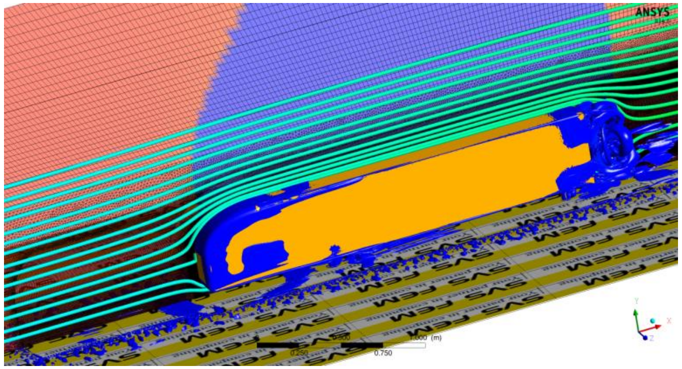




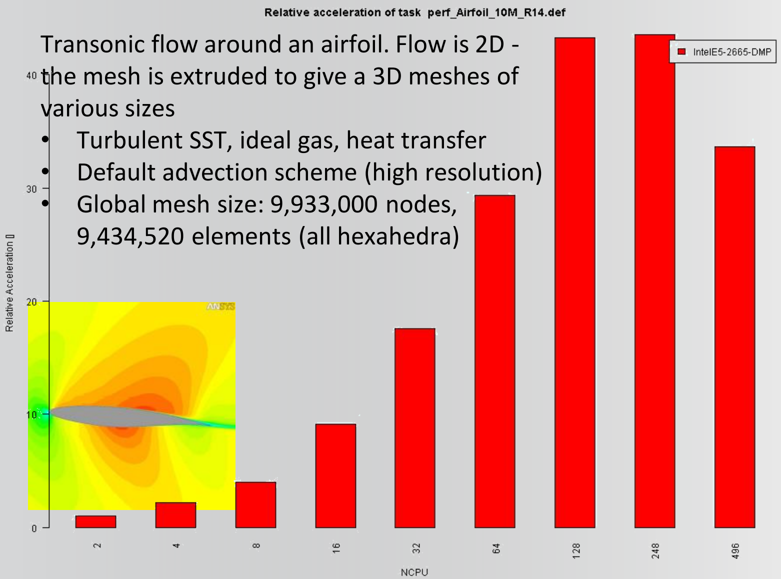
Benchmarky Fluent



111M - 3d,des,seg External flow around a truck body

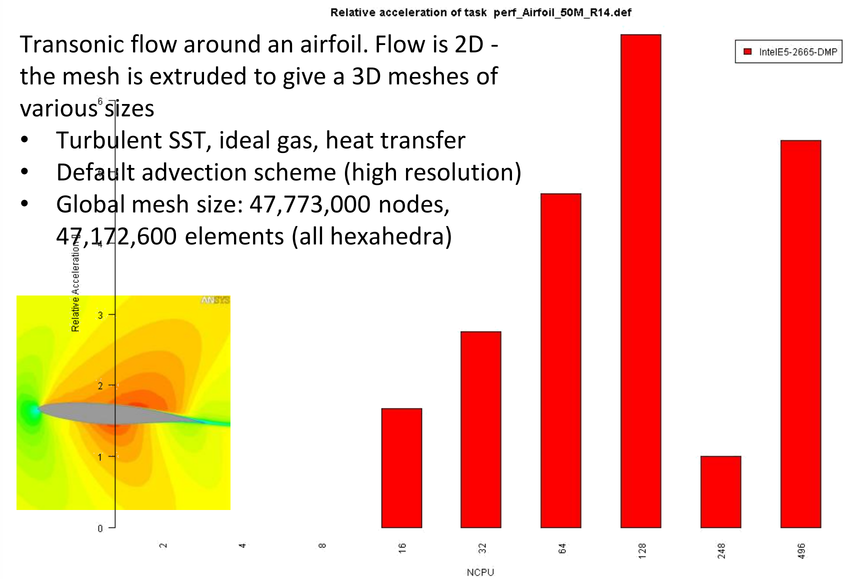


Benchmarky CFX



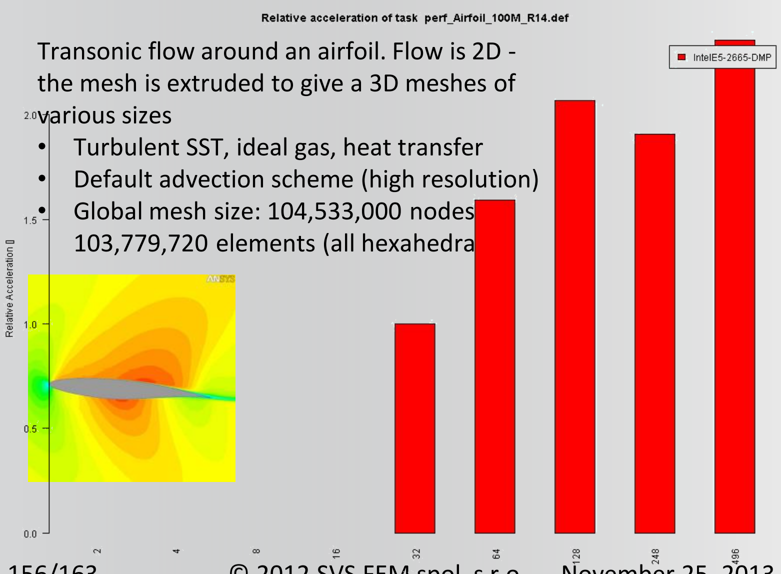
Transonic flow around an airfoil. Flow is 2D - the mesh is extruded to give a 3D meshes of various sizes

- Turbulent SST, ideal gas, heat transfer
- Default advection scheme (high resolution)
- Global mesh size: 9,933,000 nodes, 9,434,520 elements (all hexahedra)



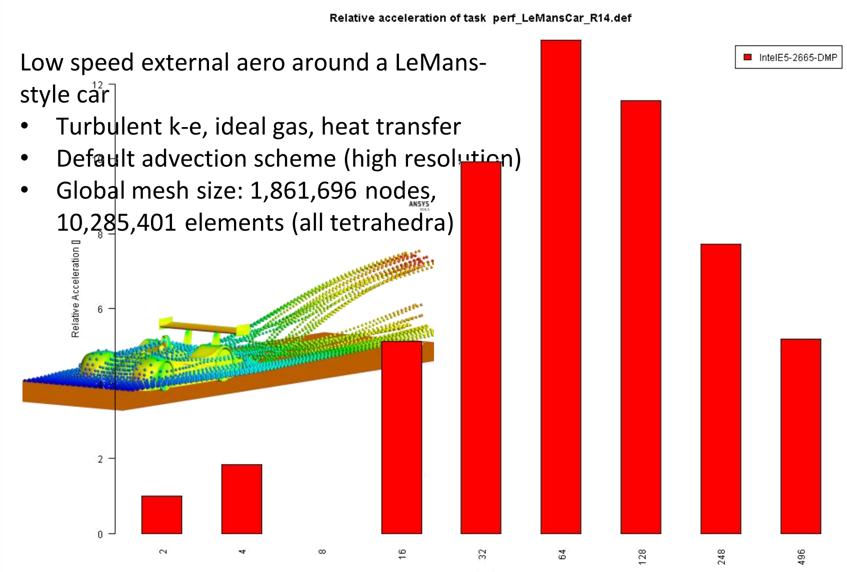
Transonic flow around an airfoil. Flow is 2D - the mesh is extruded to give a 3D meshes of various sizes

- Turbulent SST, ideal gas, heat transfer
- Default advection scheme (high resolution)
- Global mesh size: 47,773,000 nodes, 47,172,600 elements (all hexahedra)



Transonic flow around an airfoil. Flow is 2D - the mesh is extruded to give a 3D meshes of various sizes

- Turbulent SST, ideal gas, heat transfer
- Default advection scheme (high resolution)
- Global mesh size: 104,533,000 nodes, 103,779,720 elements (all hexahedra



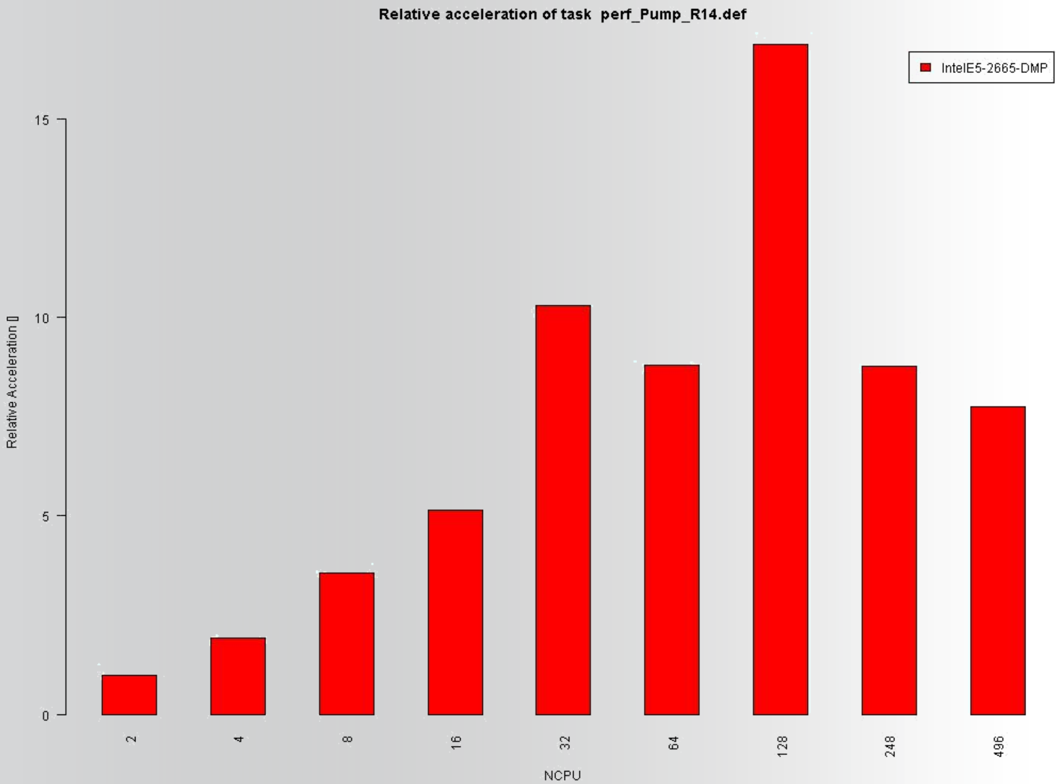
Low speed external aero around a LeMans-style car

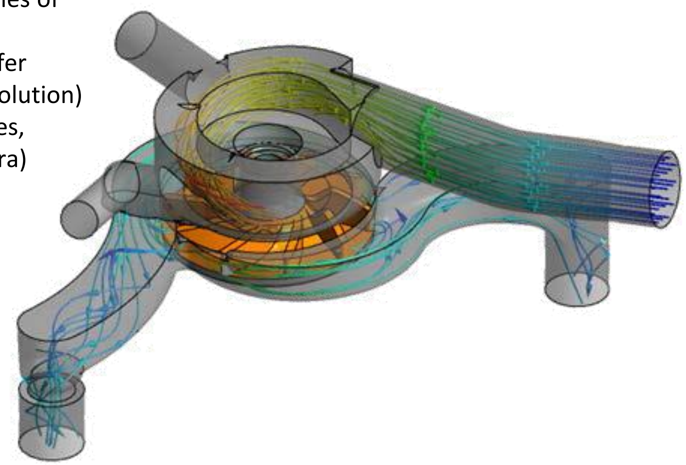
- Turbulent k-e, ideal gas, heat transfer
- Default advection scheme (high resolution)
- Global mesh size: 1,861,696 nodes, 10,285,401 elements (all tetrahedra)

Benchmarky CFX

Transonic flow around an airfoil. Flow is 2D - the mesh is extruded to give a 3D meshes of various sizes

- Turbulent SST, ideal gas, heat transfer
- Default advection scheme (high resolution)
- Global mesh size: 104,533,000 nodes, 103,779,720 elements (all hexahedra)





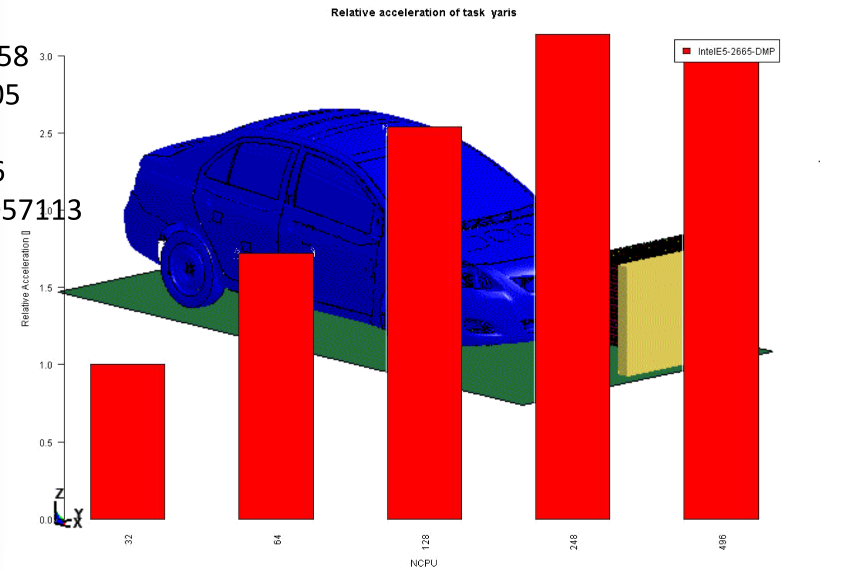
  November 25, 2013

ANSYS

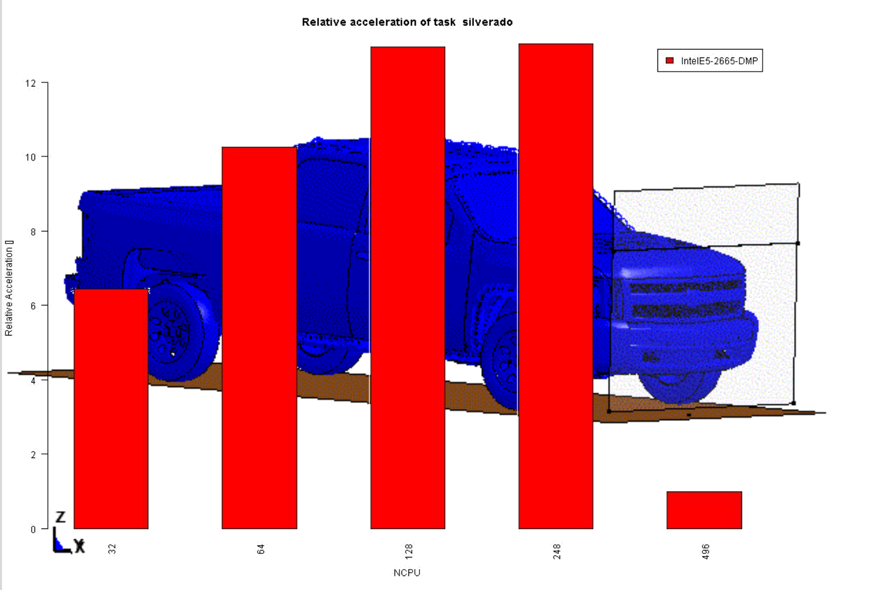
Benchmarky LS-DYNA

SVS FEM
Your partner in computing

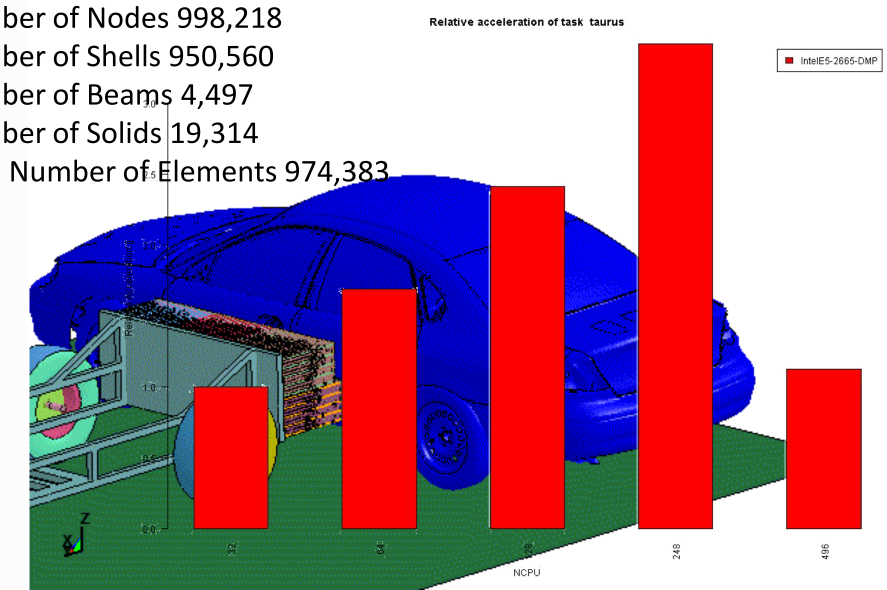
Taurus
Number of Parts - 778
Number of Nodes - 936258
Number of Shells - 805505
Number of Beams - 4
Number of Solids - 99486
Number of Elements - 1057113



Silverado
Number of Parts - 679
Number of Nodes - 942677
Number of Shells - 873144
Number of Beams - 2662
Number of Solids - 53293
Number of Elements - 929131



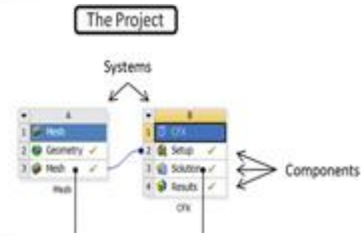
Yaris
Number of Parts 771
Number of Nodes 998,218
Number of Shells 950,560
Number of Beams 4,497
Number of Solids 19,314
Total Number of Elements 974,383



ACS infoday

SVS FEM
Your partner in computing

Znáte ACS?
3.12.2013 - Brno Hotel Avanti
vstup zdarma



ANSYS Customization Suite (ACS) Infoday
Programovací metody uživatelských úprav programového balíku ANSYS aneb „Jak si vlastnoručně doplnit co Vám v ANSYSu chybí..."

Společnost SVS FEM s.r.o. si Vás dovoluje pozvat na výjimečné setkání zaměřené na představení a názorné ukázky práce v programovacích nástrojích pro uživatelskou modifikaci prostředí ANSYS za účelem rozšíření funkcionality dle vlastní potřeby.

Kdy: **3.12. 2013 v 9 hodin.**
Kde: **Hotel Avanti**, Střední 61, Brno, 602 00
Cena: **zdarma**.

http://www.svsfem.cz/registrace

SVS FEM s.r.o.

SVS FEM s.r.o.
Škrochova 3886/42
615 00 Brno-Židenice

Tel. : +420 543 254 554, +420 543 254 555
Fax.: +420 543 254 556
Email: info@svsfem.cz
Web: http://www.svsfem.cz

Realize Your Product Promise™

Děkuji za pozornost...