



# ANSYS Structural FEA



March 2011  
**ANSYS UK**  
ANSYS, Inc. Proprietary

# The Days Agenda



AGENDA	
10:00 – 10:30	Registration & Coffee
10:30 – 10:45	Intro to Seminar & ANSYS
10:45 – 11:45	<b>Structural Mechanics Analysis using ANSYS Workbench</b> <ul style="list-style-type: none"><li>○ Overview of Workbench for Structural Analysis</li><li>○ CAD Connectivity</li><li>○ Geometry Pre-Processing</li><li>○ Meshing</li><li>○ Materials Support</li><li>○ Boundary Condition Setup</li><li>○ Analysis Types</li><li>○ Post Processing</li><li>○ Optimisation</li></ul>
11:45 – 12:15	Coffee Break
12:15 – 13:15	ANSYS FEA & Composites Guest Speaker from Even
13:15 – 13:30	Q&A Sessions.
13:30 – 14.30	Lunch & Finish

- Overview of Workbench for Structural Analysis
- CAD Connectivity
- Geometry Pre-Processing
- Meshing
- Materials Support
- Boundary Condition Setup
- Analysis Types
- Post Processing
- Optimisation



# ANSYS Workbench



# ANSYS Workbench



Unsaved Project - Workbench

File Edit View Tools Units ACP Help

New Open... Save Save As... Import... Reconnect Refresh Project Update Project Project Compact Mode

Toolbox

Project Schematic

Analysis Systems

- Electric (ANSYS)
- Explicit Dynamics (ANSYS)
- Fluid Flow - Blow Molding (POLYFLOW)
- Fluid Flow - Extrusion (POLYFLOW)
- Fluid Flow (CFX)
- Fluid Flow (FLUENT)
- Fluid Flow (POLYFLOW)
- Harmonic Response (ANSYS)
- Hydrodynamic Diffraction (AQWA)
- Linear Buckling (ANSYS)
- Magnetostatic (ANSYS)
- Modal (ANSYS)
- nCode EN Constant (DesignLife)
- nCode EN TimeSeries (DesignLife)
- nCode SN Constant (DesignLife)
- nCode SN TimeSeries (DesignLife)
- nCode SN TimeStep (DesignLife)
- nCode SN Vibration (DesignLife)
- Random Vibration (ANSYS)
- Response Spectrum (ANSYS)
- Shape Optimization (ANSYS)
- Static Structural (ANSYS)
- Steady-State Thermal (ANSYS)
- Thermal-Electric (ANSYS)
- Transient Structural (ANSYS)
- Transient Structural (MBD)
- Transient Thermal (ANSYS)

Component Systems

- AUTODYN
- BladeGen
- CFX
- CFX (Beta)
- Engineering Data
- Explicit Dynamics (LS-DYNA Export)
- External Connection
- Finite Element Modeler
- FLUENT
- Geometry
- Icepak
- Mechanical APDL
- Mechanical Model
- Mesh
- POLYFLOW
- POLYFLOW - Blow Molding
- POLYFLOW - Extrusion
- Results
- TurboGrid
- Vista TF

Custom Systems

- FSI: Fluid Flow (CFX) -> Static Structural
- FSI: Fluid Flow (FLUENT) -> Static Structural
- Pre-Stress Modal
- Random Vibration
- Response Spectrum

Project Schematic Diagram:

- A: Geometry** (1) -> **B: Mesh** (1, 2, 3, 4)
- B: Mesh** (1, 2, 3, 4) -> **C: CFX** (1, 2, 3, 4)
- B: Mesh** (1, 2, 3, 4) -> **E: FLUENT** (1, 2, 3)
- C: CFX** (1, 2, 3, 4) -> **D: Static Structural (ANSYS)** (1, 2, 3, 4, 5, 6, 7)
- E: FLUENT** (1, 2, 3) -> **F: Steady-State Thermal (ANSYS)** (1, 2, 3, 4, 5, 6, 7)
- A: Geometry** (1) -> **G: Static Structural (ANSYS)** (1, 2, 3, 4, 5, 6, 7, 8)
- G: Static Structural (ANSYS)** (1, 2, 3, 4, 5, 6, 7, 8) -> **H: nCode EN TimeSeries (DesignLife)** (1, 2, 3, 4)

Files

Name	Cell ID	Size	Type	Date Modified	Location
material.engd	G2,H2	18 KB	Engineering Data File	08/11/2010 16:14:04	C:\Users\ymitchel\AppData\Local\Temp\WB_MILRMITCHELL1_6180_2\unsaved_project_files\dp0\SYS-3\ENGD
SYS-3.engd	G4	18 KB	Engineering Data File	08/11/2010 16:14:04	C:\Users\ymitchel\AppData\Local\Temp\WB_MILRMITCHELL1_6180_2\unsaved_project_files\dp0\global\MECH
Geom.agdb	A2,B2,G3	13 KB	Geometry File	08/11/2010 16:14:05	C:\Users\ymitchel\AppData\Local\Temp\WB_MILRMITCHELL1_6180_2\unsaved_project_files\dp0\Geom\DM

Ready

Show Progress Show 2 Messages

16:16 08/11/2010

# ANSYS Workbench



Unsaved Project - Workbench

File Edit View Tools Units ACP Help

New Open... Save Save As... Import... Reconnect Refresh Project Update Project Project Compact Mode

**Analysis Systems**

- Electric (ANSYS)
- Explicit Dynamics (ANSYS)
- Fluid Flow - BlowMolding (POLYFLOW)
- Fluid Flow - Extrusion (POLYFLOW)
- Fluid Flow (CFX)
- Fluid Flow (FLUENT)
- Fluid Flow (POLYFLOW)
- Harmonic Response (ANSYS)
- Hydrodynamic Diffraction (AQWA)
- Linear Buckling (ANSYS)
- Magnetostatic (ANSYS)
- Modal (ANSYS)
- nCode EN Constant (DesignLife)
- nCode EN TimeSeries (DesignLife)
- nCode SN Constant (DesignLife)
- nCode SN TimeSeries (DesignLife)
- nCode SN TimeStep (DesignLife)
- nCode SN Vibration (DesignLife)
- Random Vibration (ANSYS)
- Response Spectrum (ANSYS)
- Shape Optimization (ANSYS)
- Static Structural (ANSYS)
- Steady-State Thermal (ANSYS)
- Thermal-Electric (ANSYS)
- Transient Structural (ANSYS)
- Transient Structural (MBD)
- Transient Thermal (ANSYS)

The diagram shows the following analysis systems and their dependencies:

- B: Mesh** (1. Mesh, 2. Geometry, 3. Mesh, 4. Parameters)
- C: CFX** (1. CFX, 2. Setup, 3. Solution, 4. Results) - depends on B
- D: Static Structural (ANSYS)** (1. Static Structural (ANSYS), 2. Engineering Data, 3. Geometry, 4. Model, 5. Setup, 6. Solution, 7. Results) - depends on B and C
- E: FLUENT** (1. FLUENT, 2. Setup, 3. Solution) - depends on B
- F: Steady-State Thermal (ANSYS)** (1. Steady-State Thermal (ANSYS), 2. Engineering Data, 3. Geometry, 4. Model, 5. Setup, 6. Solution, 7. Results) - depends on E
- G: Static Structural (ANSYS)** (1. Static Structural (ANSYS), 2. Engineering Data, 3. Geometry, 4. Model, 5. Setup, 6. Solution, 7. Results, 8. Parameters) - depends on B
- H: nCode EN TimeSeries (DesignLife)** (1. nCode EN TimeSeries (DesignLife), 2. Engineering Data, 3. Solution, 4. Results) - depends on G

No data

No data	
A	B
Property	Value

ID	Size	Type	Date Modified	Location
H2	18 KB	Engineering Data File	08/11/2010 16:14:04	C:\Users\ymitchel\AppData\Local\Temp\WB_MILRMITCHELL1_6180_2\unsaved_project_files\dp0\SYS3\ENGD
	18 KB	Engineering Data File	08/11/2010 16:14:04	C:\Users\ymitchel\AppData\Local\Temp\WB_MILRMITCHELL1_6180_2\unsaved_project_files\dp0\global\MECH
B2,G3	13 KB	Geometry File	08/11/2010 16:14:05	C:\Users\ymitchel\AppData\Local\Temp\WB_MILRMITCHELL1_6180_2\unsaved_project_files\dp0\Geom\DM

Ready

Show Progress Show 2 Messages

16:16 08/11/2010

# ANSYS Workbench



The screenshot displays the ANSYS Workbench interface with a project workflow diagram. The workflow consists of the following components:

- Component A: Geometry** (DM)
  - 1. Geometry (DM)
  - 2. Geometry (DM) [checked]
  - 3. Parameters
- Component B: Mesh**
  - 1. Mesh
  - 2. Geometry (DM) [checked]
  - 3. Mesh
  - 4. Parameters
- Component C: CFX**
  - 1. CFX
  - 2. Setup
  - 3. Solution
  - 4. Results
- Component D: Static Structural (ANSYS)**
  - 1. Static Structural (ANSYS)
  - 2. Engineering Data [checked]
  - 3. Geometry
  - 4. Model
  - 5. Setup
  - 6. Solution
  - 7. Results
- Component E: FLUENT**
  - 1. FLUENT
  - 2. Setup
  - 3. Solution
- Component F: Steady-State Thermal (ANSYS)**
  - 1. Steady-State Thermal (ANSYS)
  - 2. Engineering Data [checked]
  - 3. Geometry
  - 4. Model
  - 5. Setup
  - 6. Solution
  - 7. Results
- Component G: Static Structural (ANSYS)**
  - 1. Static Structural (ANSYS)
  - 2. Engineering Data [checked]
  - 3. Geometry [checked]
  - 4. Model [checked]
  - 5. Setup [checked]
  - 6. Solution [checked]
  - 7. Results [checked]
  - 8. Parameters
- Component H: nCode EN TimeSeries (DesignLife)**
  - 1. nCode EN TimeSeries (DesignLife)
  - 2. Engineering Data [checked]
  - 3. Solution
  - 4. Results

The workflow is connected as follows: Component A feeds into Component B. Component B feeds into Component C and Component D. Component C feeds into Component E. Component E feeds into Component F. Component F feeds into Component G. Component G feeds into Component H. A red arrow points from Component A to Component G. A red arrow points from Component B to Component G. A red arrow points from Component G to Component H.

**Files Panel:**

Name	Cell ID	Size	Type	Date Modified	Location
material.engd	G2,H2	18 KB	Engineering Data File	08/11/2010 16:14:04	C:\Users\ymitchel\AppData\Local\Temp\WB_MILRMITCHELL1_6180_2\unsaved_project_files\dp0\SYS-3\ENG.D
SYS-3.engd	G4	18 KB	Engineering Data File	08/11/2010 16:14:04	C:\Users\ymitchel\AppData\Local\Temp\WB_MILRMITCHELL1_6180_2\unsaved_project_files\dp0\global\MECH
Geom.agdb	A2,B2,G3	13 KB	Geometry File	08/11/2010 16:14:05	C:\Users\ymitchel\AppData\Local\Temp\WB_MILRMITCHELL1_6180_2\unsaved_project_files\dp0\Geom\DM

The bottom of the screen shows the Windows taskbar with the system clock at 16:16 on 08/11/2010.

# ANSYS Workbench



The screenshot displays the ANSYS Workbench interface. At the top, the title bar reads "Unsaved Project - Workbench". The menu bar includes File, Edit, View, Tools, Units, ACP, and Help. Below the menu bar is a toolbar with icons for New, Open..., Save, Save As..., Import..., Reconnect, Refresh Project, Update Project, Project, and Compact Mode. The main workspace is divided into several panes:

- Toolbox:** A vertical list of analysis systems including Electric (ANSYS), Explicit Dynamics (ANSYS), Fluid Flow - Blow Molding (POLYFLOW), Fluid Flow - Extrusion (POLYFLOW), Fluid Flow (CFX), Harmonic, Hydro, Linear, Magnetic, Modal, Nonlinear, Nonlinear Buckling, Nonlinear Dynamic, Nonlinear Static, Random Vibration, Response Spectrum, Static, Steady State, Thermal, Transient, Transient Buckling, Transient Dynamic, Transient Nonlinear, Transient Thermal, and Transient Viscoplasticity.
- Project Schematic:** A central area showing a workflow from left to right. It includes a "Parameter Set" at the top, followed by four analysis systems: A (Geometry), B (Mesh), C (CFX), and D (Static Structural (ANSYS)).
- Optimization Methods View:** A detailed view of optimization methods, showing a "Parameter Set" at the top connected to three main optimization categories: I (Response Surface), J (Six Sigma Analysis), and K (Goal Driven Optimization). Below these are L (Direct Optimization). Each category has a list of sub-methods: I includes Response Surface, Design of Experiments, and Response Surface; J includes Six Sigma Analysis, Design of Experiments (SSA), Response Surface (SSA), and Six Sigma Analysis; K includes Goal Driven Optimization, Design of Experiments, Response Surface, and Optimization; L includes Direct Optimization (Beta) and Optimization. Blue arrows indicate dependencies between these methods.
- Custom Systems:** A table at the bottom left listing custom systems with columns for Name, ID, Size, File Type, Date, and Path.

Custom Systems	ID	Size	File Type	Date	Path	
FSI: Fluid Flow (CFX) -> Static Structural	84	16 KB	Engineering Data File	08/11/2010 16:14:04	C:\Users\ymitchel\AppData\Local\Temp\WB_MILRMITCHELL1_6180_2\unsaved_project_files\dp0\global\...	
FSI: Fluid Flow (FLUENT) -> Static Structural	4	A2,B2,G3	13 KB	Geometry File	08/11/2010 16:14:05	C:\Users\ymitchel\AppData\Local\Temp\WB_MILRMITCHELL1_6180_2\unsaved_project_files\dp0\Geom\DM

The bottom status bar shows "Ready" and "Show Progress" / "Show 2 Messages". The Windows taskbar at the very bottom shows the system clock as 16:16 on 08/11/2010.





# ANSYS Mechanical Analysis steps



- **Geometry**

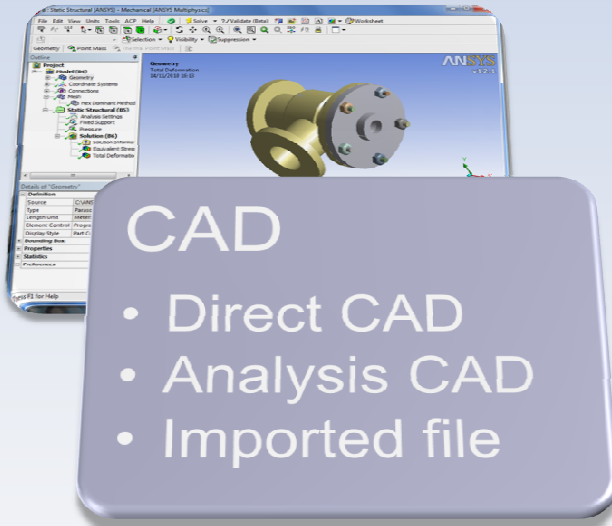
- Direct CAD Links

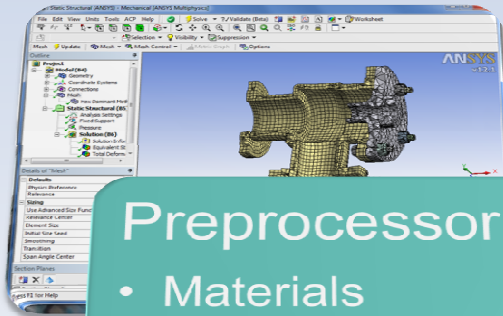
- Connect to real CAD models and create true parametric analysis

- Create analysis geometry

- Geometry clean-up
    - Simplification
    - Create Shell & Beam geometry

- Work with imported files



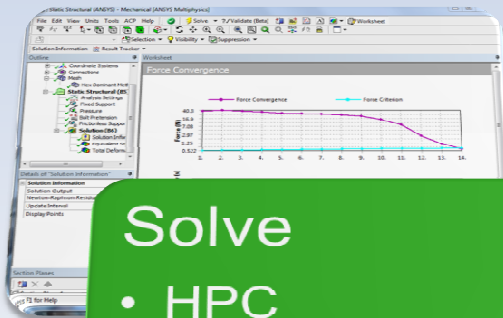


## Preprocessor

- Materials
- Meshing
- Loads
- Contact

- **Preprocessing**
  - **Materials**
    - Linear-Elastic
    - Plastic
    - Hyper-elastic
    - Creep
    - Soils, Concrete
    - Damage models
  - **Meshing**
    - From fully automatic to highly controlled
  - **Loads**
    - Imported data fields
    - Time dependant
    - Complex systems
  - **Contact**
    - Model real assemblies
    - Bonded, Frictionless & Frictional contact

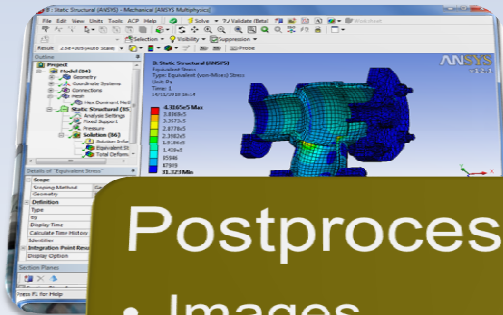
- Solving
  - ANSYS solver technology evolving to keep pace with PC developments
  - Multi-core
  - 32 & 64 bit
  - Clusters
  - GPU



Solve

- HPC
- Non-linearities
- Accuracy

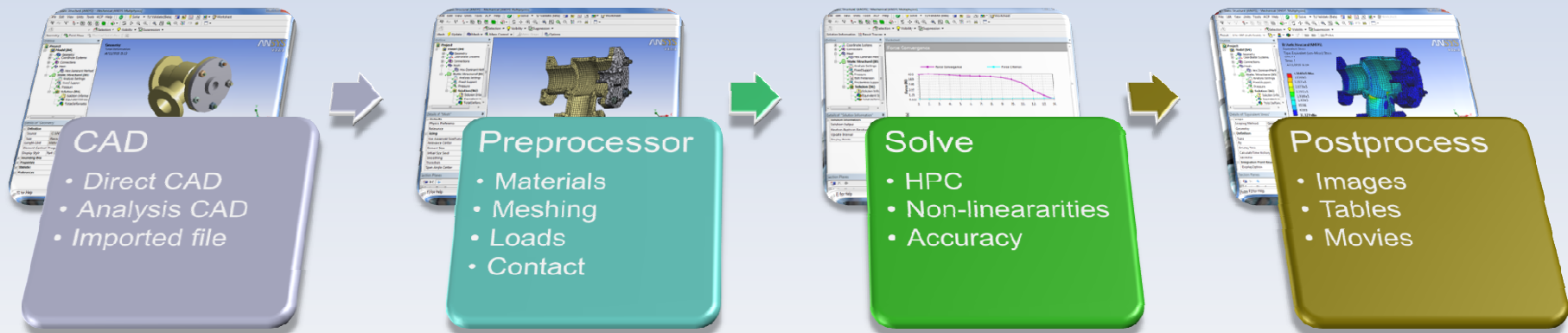
- **Postprocessing**
  - **Stress, Strain, Creep, Contact, Reactions**
  - **Linearisation**
  - **Images**
  - **Tabular data → Excel**
  - **Movie files**
  - **Automated report generation**



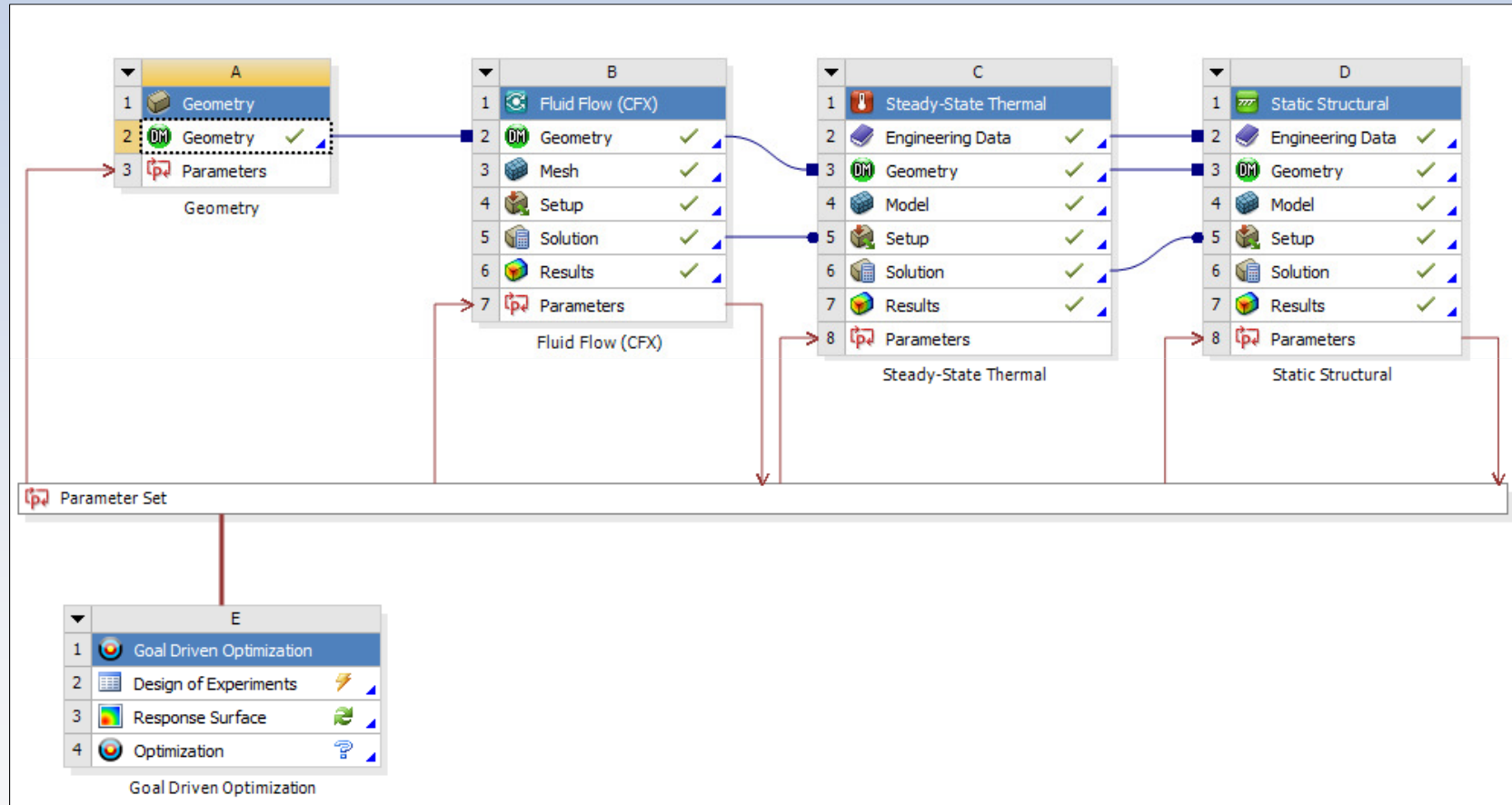
## Postprocess

- Images
- Tables
- Movies

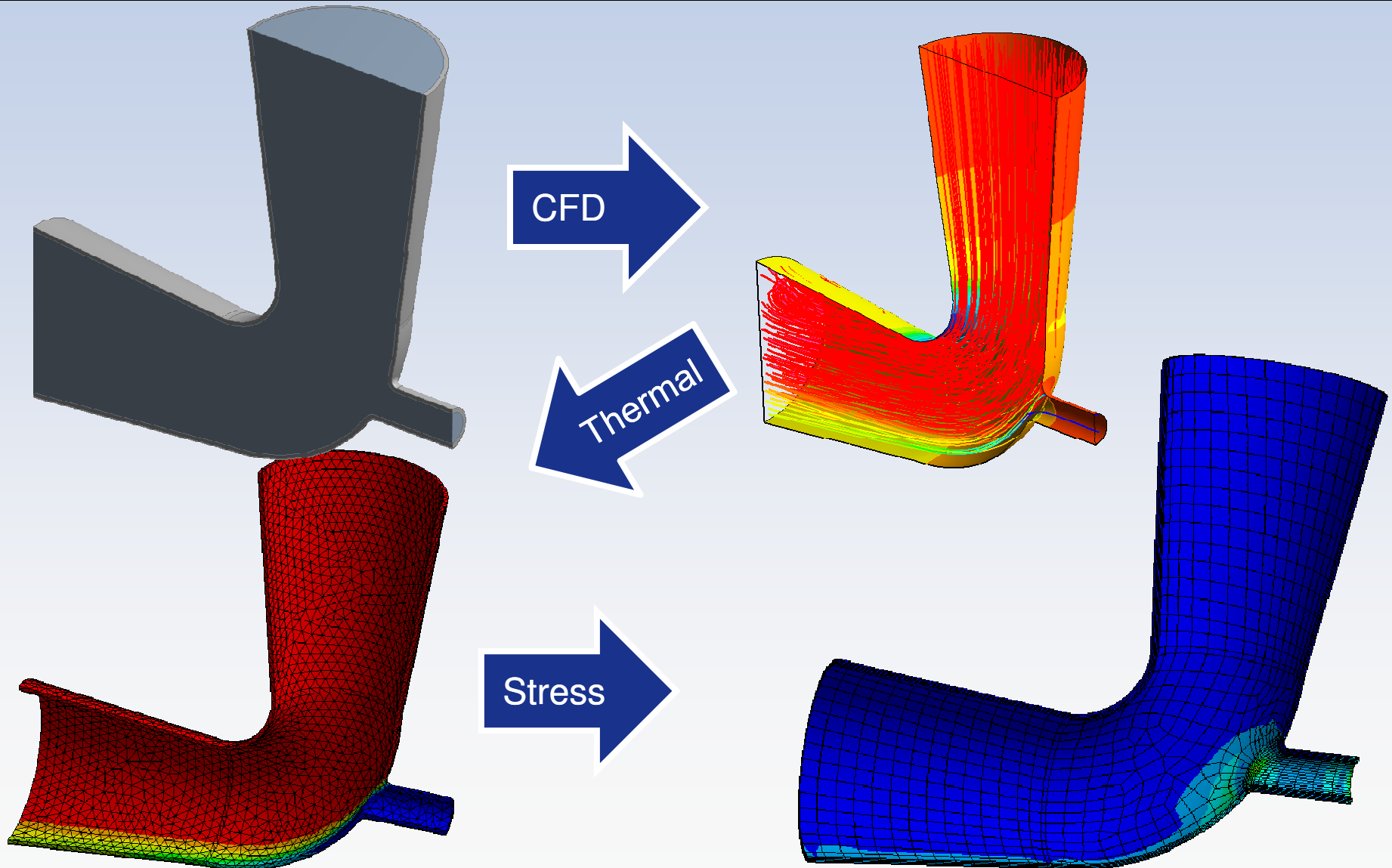
## • ANSYS structural analysis



# Example workflow



# Linked analysis



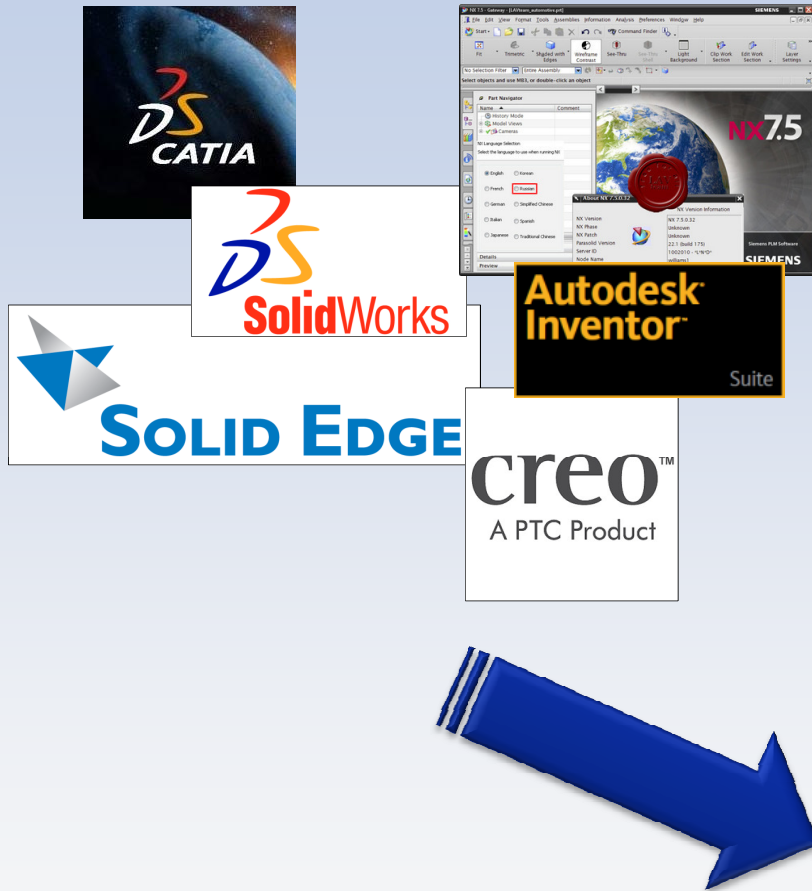




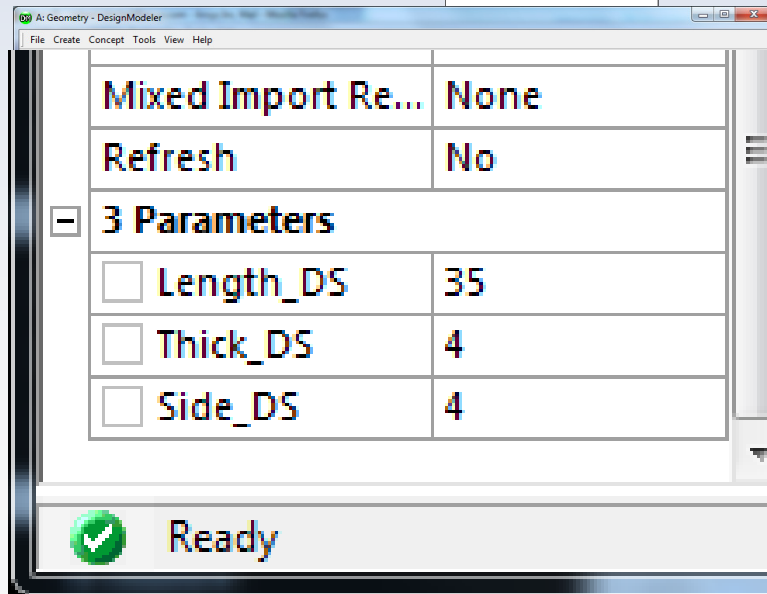
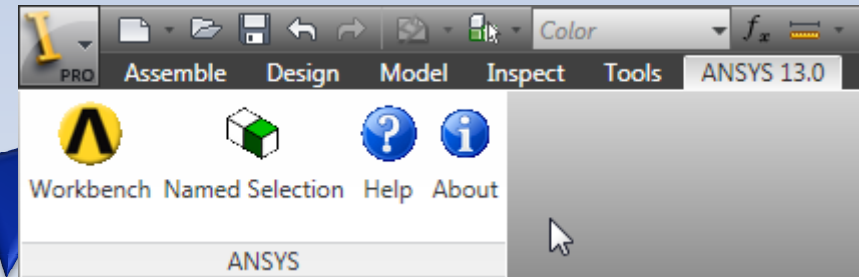
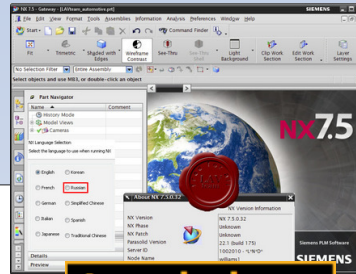
## ANSYS Geometry and CAD options



# Bi-Directional Integration Interfaces



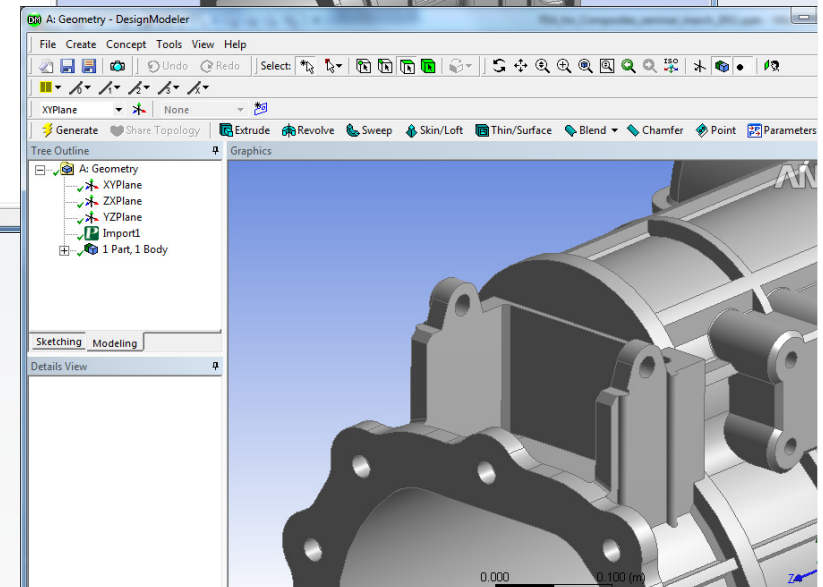
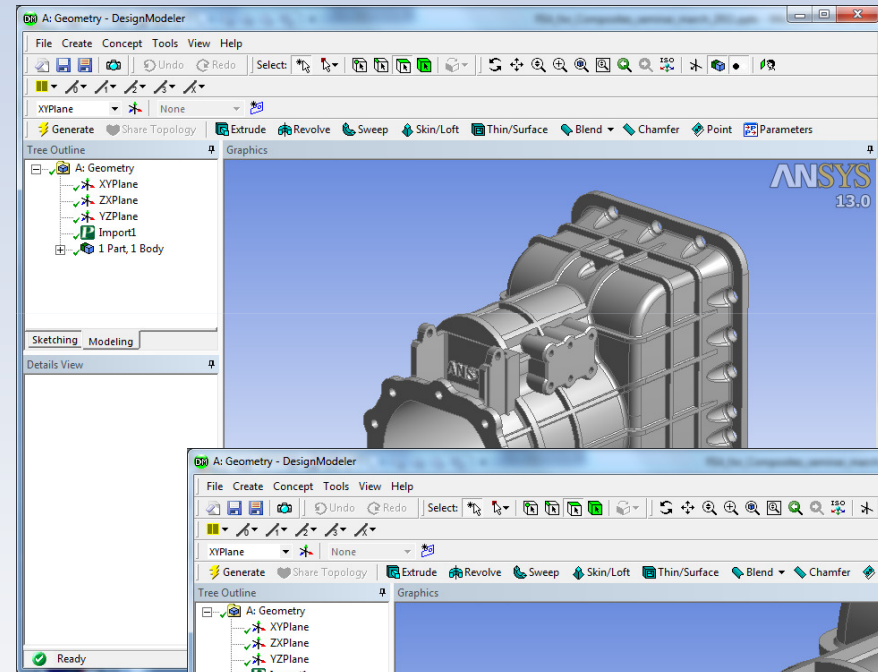
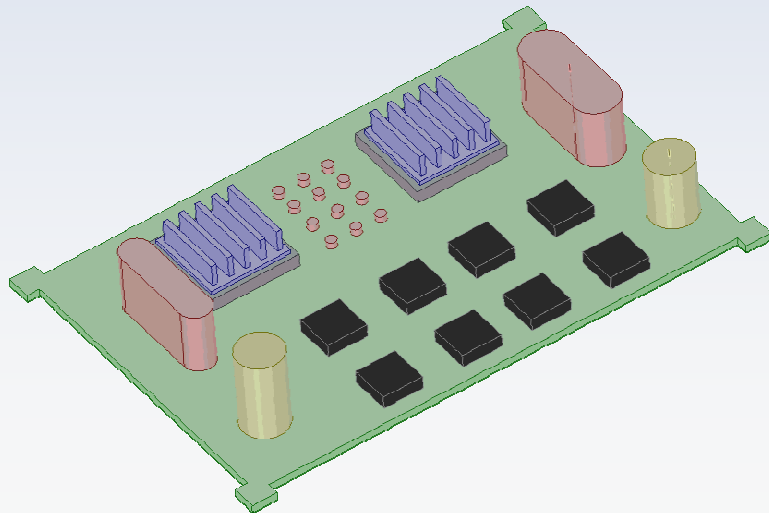
# Bi-Directional geometric interfaces



# Geometry editing tools



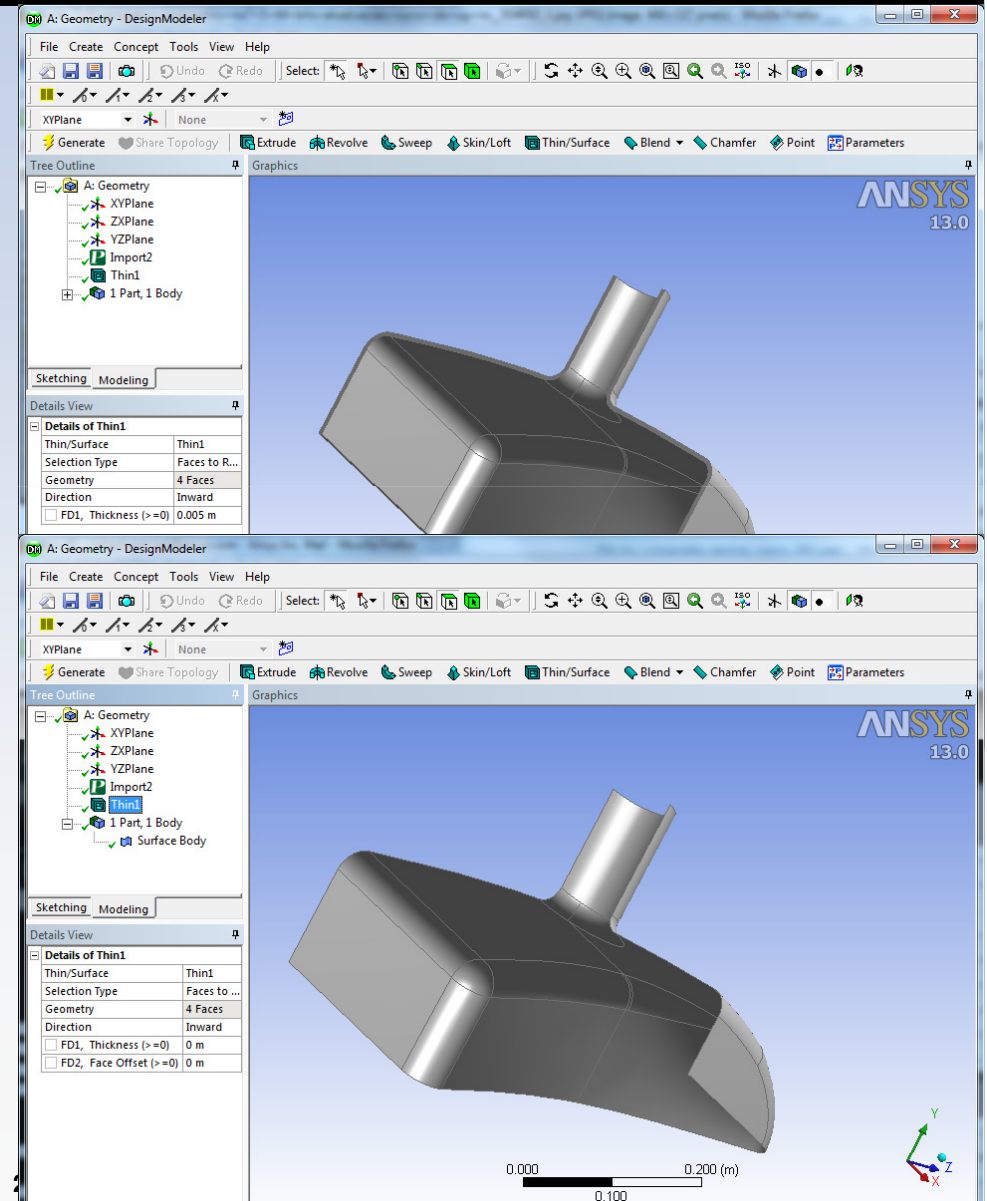
- Turning CAD geometry into analysis geometry.
- Remove small features
- Add parameters



# Mid-surfacing



- ANSYS ACP works on shell (surface) geometry.
- DesignModeler & Space Claim both have tools for turning “thick/solid” geometry into surface geometry.





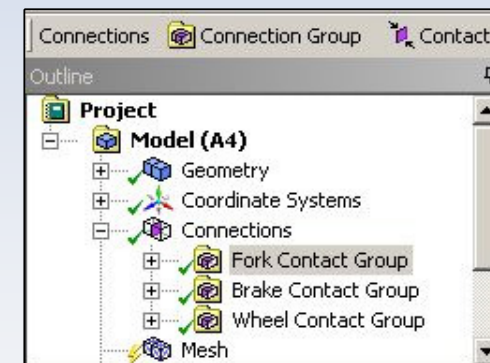
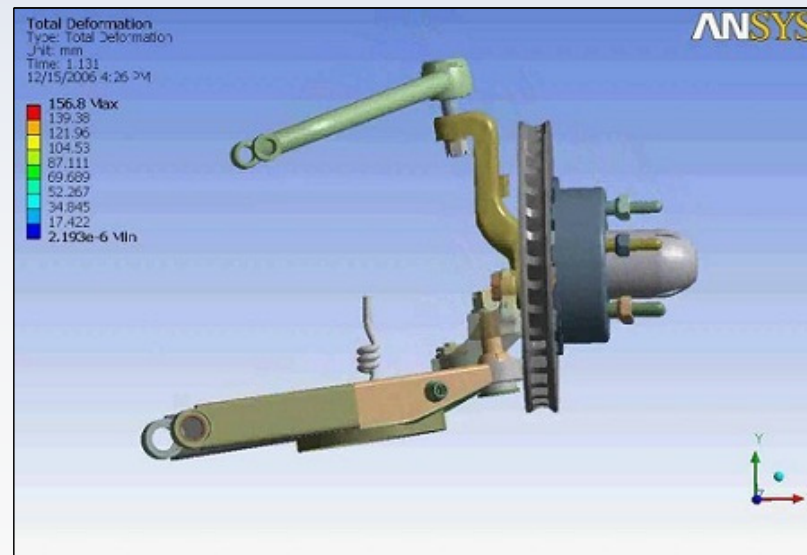
# ANSYS Contact



# Connections



- Automatic contact detection
- Contact/Spot welds to define interaction
- Number of contact definitions/formulations available
- Contacts can be grouped



# Contact types



- Realistic behaviour only possible with advanced contact.
- Optimised settings for each contact option
- Range of formulations
- Contact types
  - Bonded
  - No Separation
  - Frictionless
  - Rough
  - Frictional

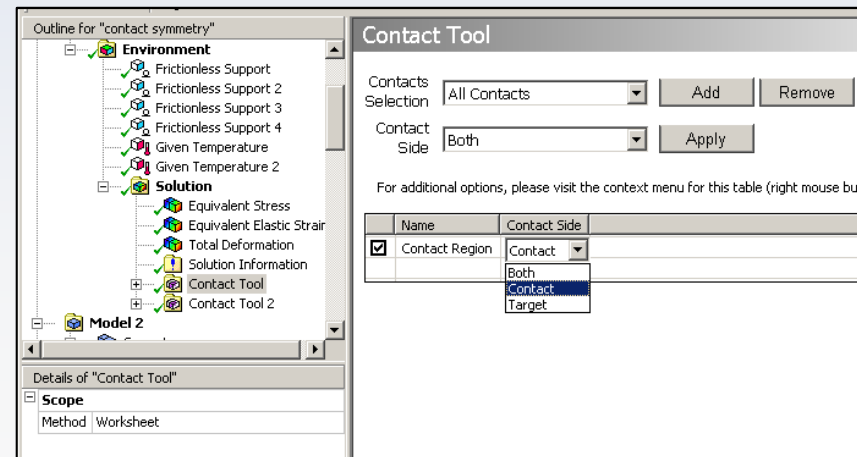
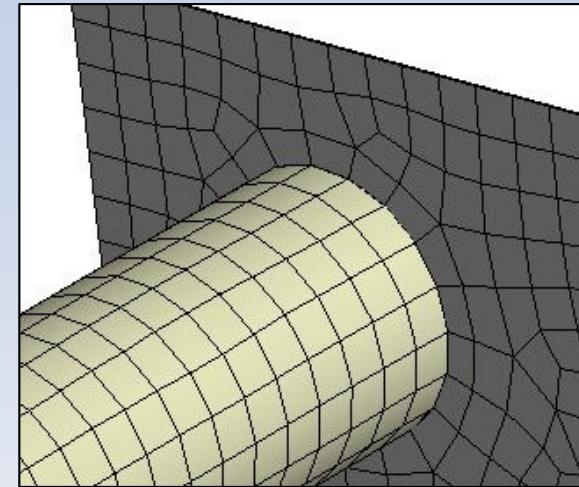
[-] Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	1 Face
Contact Bodies	Part 4
Target Bodies	Solid
[-] Definition	
Type	Bonded
Scope Mode	Bonded
Behavior	No Separation
Trim Contact (Beta)	Frictionless
Suppressed	Rough
	Frictional
[-] Advanced	
Formulation	Pure Penalty
Normal Stiffness	Program Controlled
Update Stiffness	Never
Pinball Region	Program Controlled



# Connections



- **Mesh connections**
  - Connect shell geometry up at a mesh level
- **Pre and post contact tool**
  - Plot pressures
  - Contact status





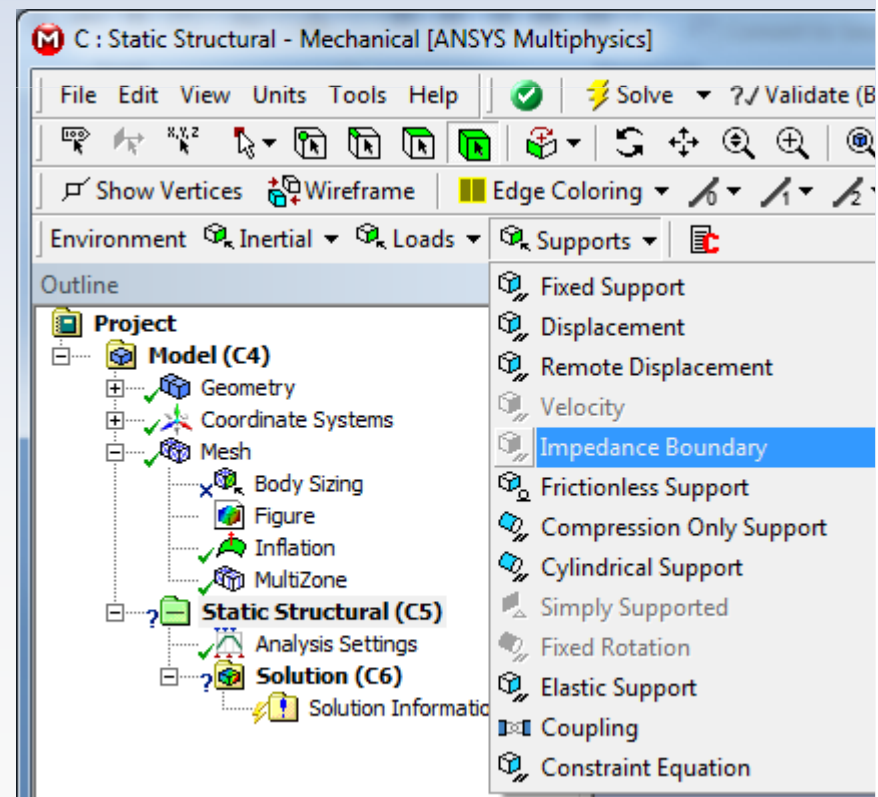
# ANSYS Pre-processing



# Boundary conditions



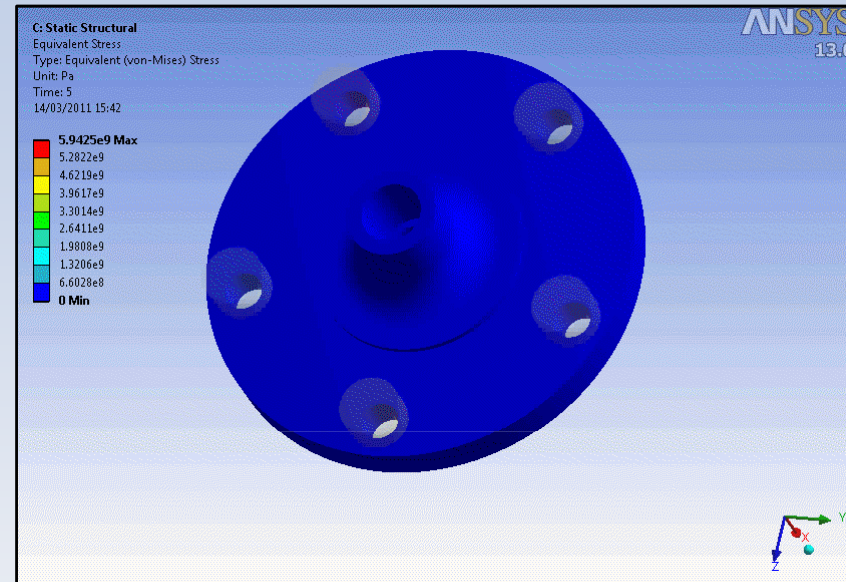
- Extensive list of constraints and loads within the toolbar
- Ability to apply loadings from CFD
- Ability to import loadings from text file
- Persistent application

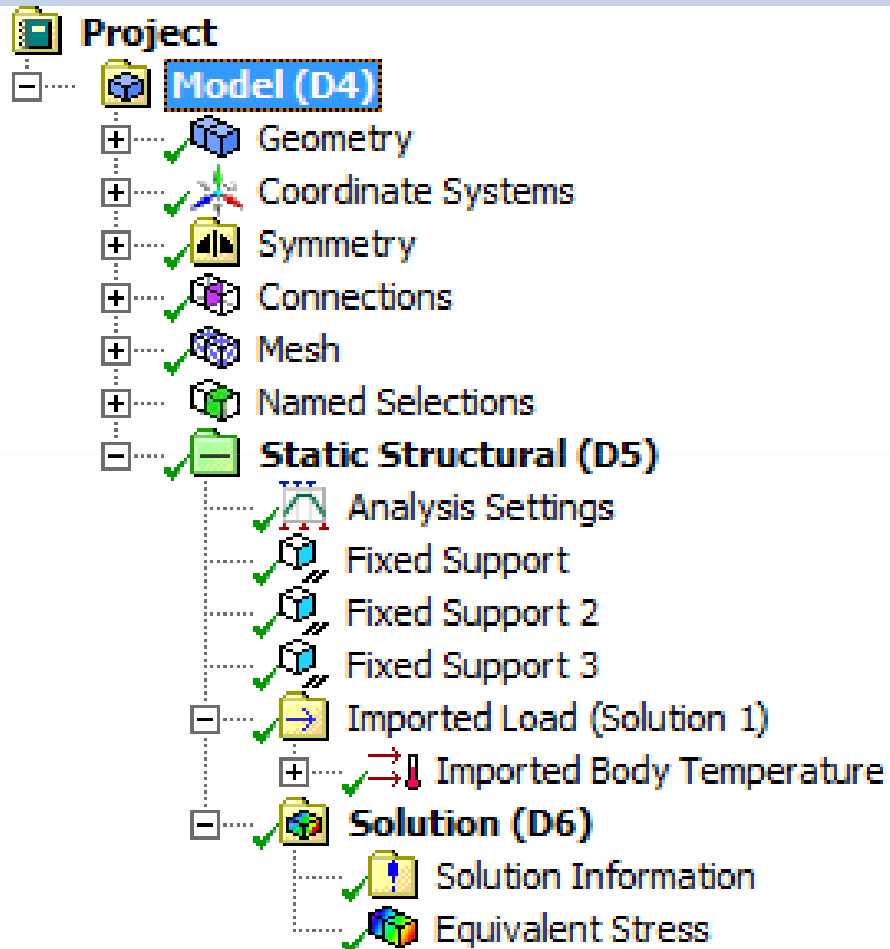


# Analysis types



- **Static Structural**
- **Modal**
- **Harmonic**
- **Buckling**
- **Random Vibration**
- **Response Spectrum**
- **Transient Structural**
- **Thermal**
- **Transient Thermal**





- The tools in Workbench are geared to allow engineers to carry out **ENGINEERING**
- Complex tasks made intuitive



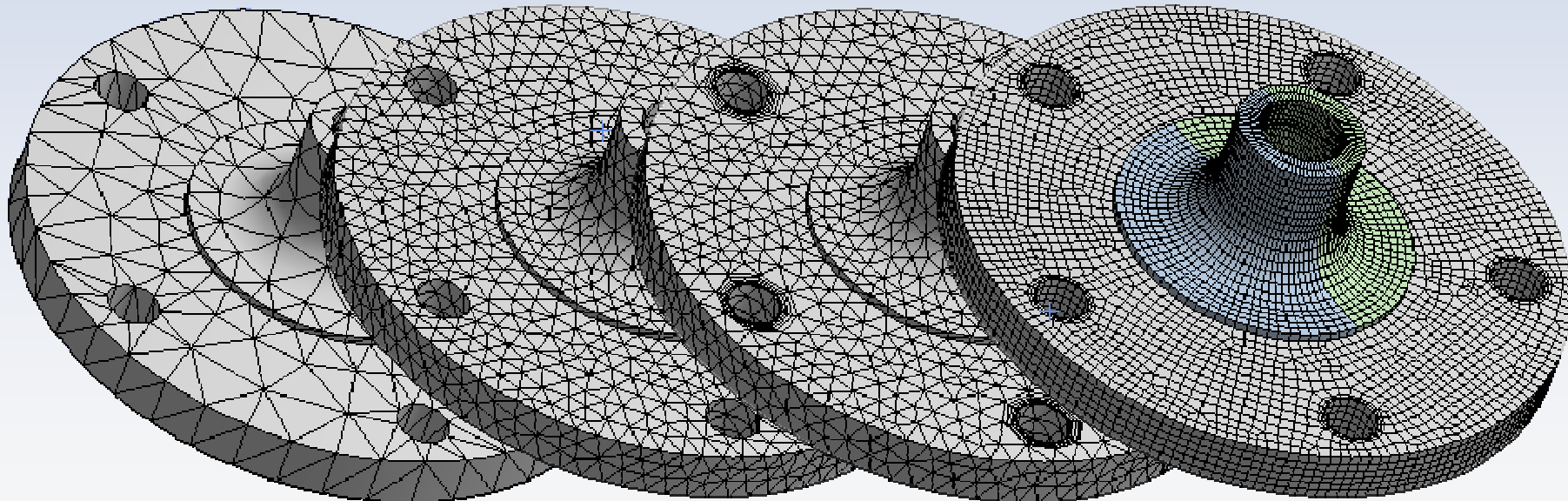
# ANSYS Meshing



# Meshing options



- **ANSYS Meshing has options from fully automatic to highly controlled.**





## ANSYS Core technology





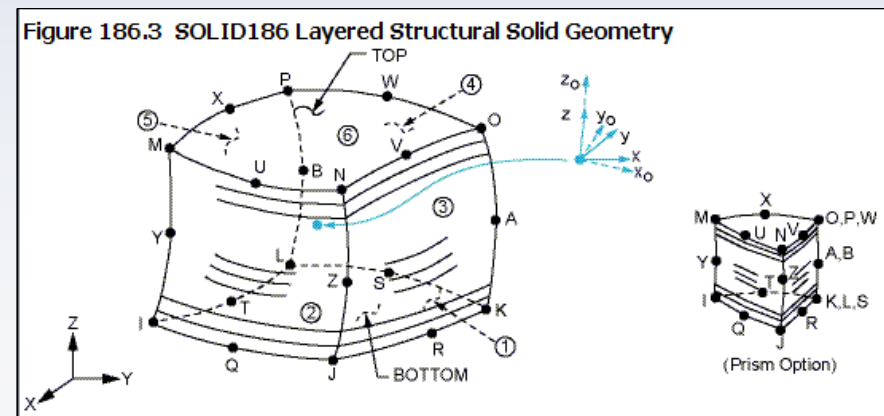
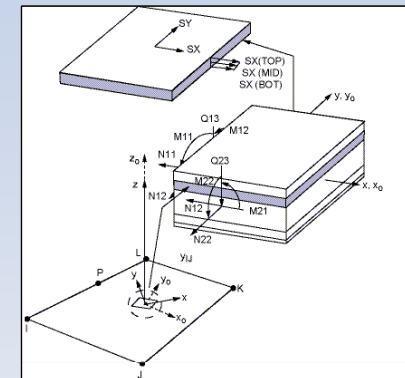
- **Elements**

- Range of elements for solids

- Tets, pyramids & hex
- Coupled physics
- Low and high order

- Shell elements

- Low and high order
- Layered options



**Create and Modify Shell Sections**

Section Edit Tools

Layup Section Controls Summary

Layup

Create and Modify Shell Sections Name:  ID:

	Thickness	Material ID	Orientation	Integration Pts	Pictorial View
4	<input type="text" value="0.65"/>	<input type="text" value="2"/>	<input type="text" value="-45"/>	<input type="text" value="5"/>	
3	<input type="text" value="0.45"/>	<input type="text" value="1"/>	<input type="text" value="90"/>	<input type="text" value="3"/>	
2	<input type="text" value="0"/>	<input type="text" value="2"/>	<input type="text" value="30"/>	<input type="text" value="3"/>	
1	<input type="text" value="0.34"/>	<input type="text" value="1"/>	<input type="text" value="45"/>	<input type="text" value="5"/>	

Section Offset:  User Defined Value:

Section Function:  KCN or Node:

ANSYS Mechanical Utility Menu

File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help

ANSYS Toolbar

ANSYS Main Menu

- Preferences
  - Preprocessor
    - Element Type
    - Real Constants
    - Material Props
    - Sections
      - Section Library
        - Beam
        - Shell
          - Lay-up
            - Add / Edit
            - Plot Section
            - Pre-integrated
          - Pretension
          - Joints
          - Reinforcement
          - List Sections
          - Delete Section
- Modeling
- Meshing
- Checking Ctrls
- Numbering Ctrls
- Archive Model
- Coupling / Ceqn
- Multi-field Set Up
- Loads
- Physics
- Path Operations

- Solution
- General Postproc
- TimeHist Postpro
- Topological Opt
- Design Opt
- Prob Design
- Radiation Opt
- Run-Time Stats

Section Editor

```

1
LAYER STACKING
ELEM      =    0
SECT      =    1
LAYERS    :
TOTAL     =    4
SHOWN     :
FROM      1 TO 4
    
```

ANSYS  
OCT 9 2006  
12:04:01

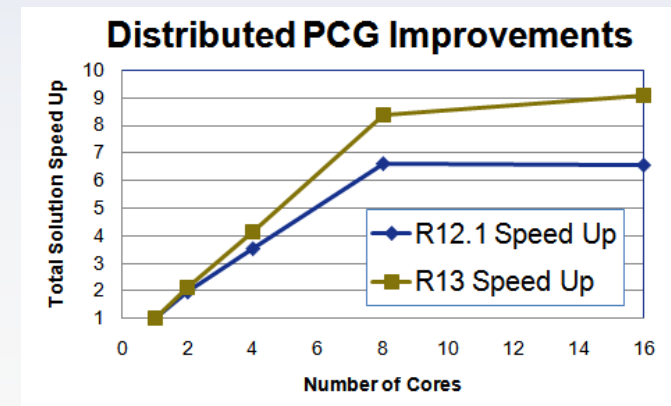
Layer#	Material#
1	1
2	2
3	1
4	2

Theta  
45  
30  
90  
-45

Pick a menu item or enter an ANSYS Command (PREP7)    mat=1    type=1    real=1    csys=0    secn=1

## Solver technology

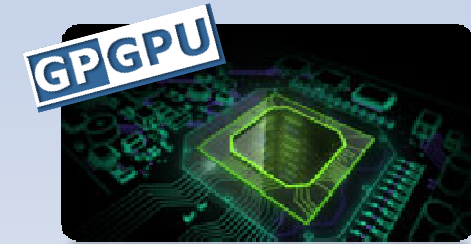
- **Solvers work with your hardware**
  - Workstations now ship with multiple cores
  - Solvers evolved to make best use of this
  - Remote solving options
- **Enhanced scalability**
  - Many enhancements to improve performance of DPCG and DSPARSE solvers
- **64bit native code**
  - Faster solve times



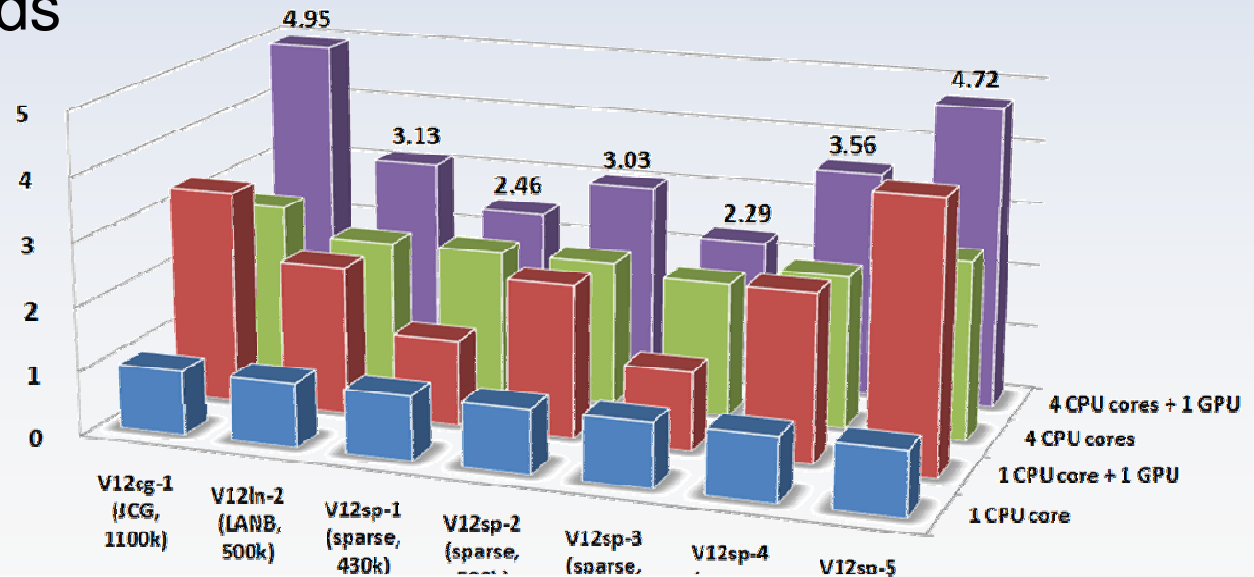
# GPU Accelerator Feature



- SMP SPARSE and PCG solvers only
- Activated with `-acc @` command line
- Supported on Windows/Linux 64-bit systems
- Currently **only available for nVidia Tesla 20-series (1U) cards**



- Intel Xeon 5560 (2.8 GHz, 8 cores total)
- 32 GB of RAM
- Windows XP SP2 (64-bit)
- Tesla C2050



Overall Simulation Speedups for R12 Benchmark Set

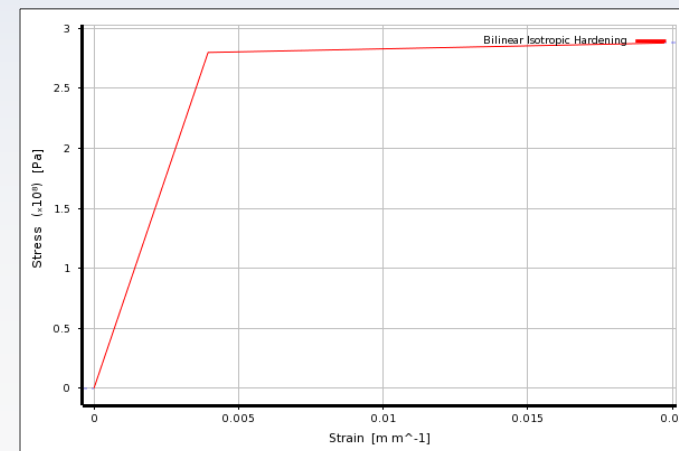


# ANSYS Materials



- Accurate materials are an essential requirement for any analysis
- ANSYS has a wide range available to engineers.
- User definable material libraries.

Engineering Data Sources	
A	
1	Data Source
2	★ Favorites
3	📁 General Materials
4	📁 General Non-linear Materials
5	📁 Explicit Materials
6	📁 Hyperelastic Materials
7	📁 Magnetic B-H Curves
8	📁 Thermal Materials



# ANSYS materials continued



- **Metallic**

- Linear
- Plasticity
- Temperature effect
- Creep
- Test data

Low deflection, room temperature

Large deflection, permanent deformation

Thermal loads

Long timescale effects

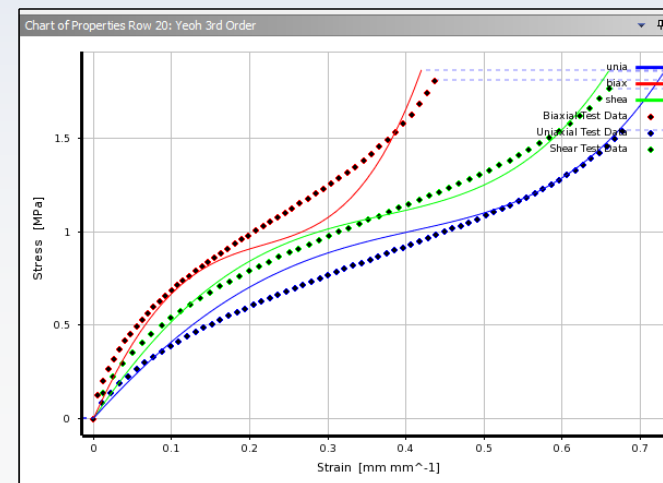
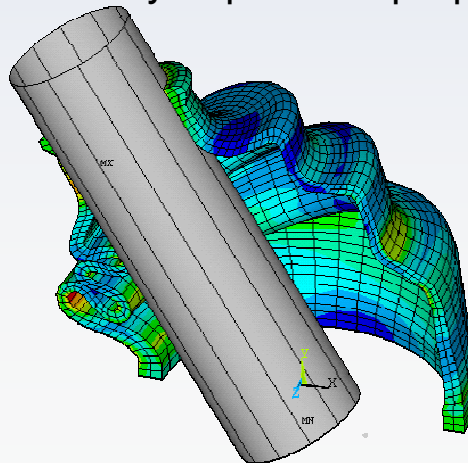
Can be used directly

- **Specialist materials**

- Hyper-elasticity
- Concretes & soils
- Directionally dependant properties

Rubbers, polymers etc

Crushing and brittle failure modes

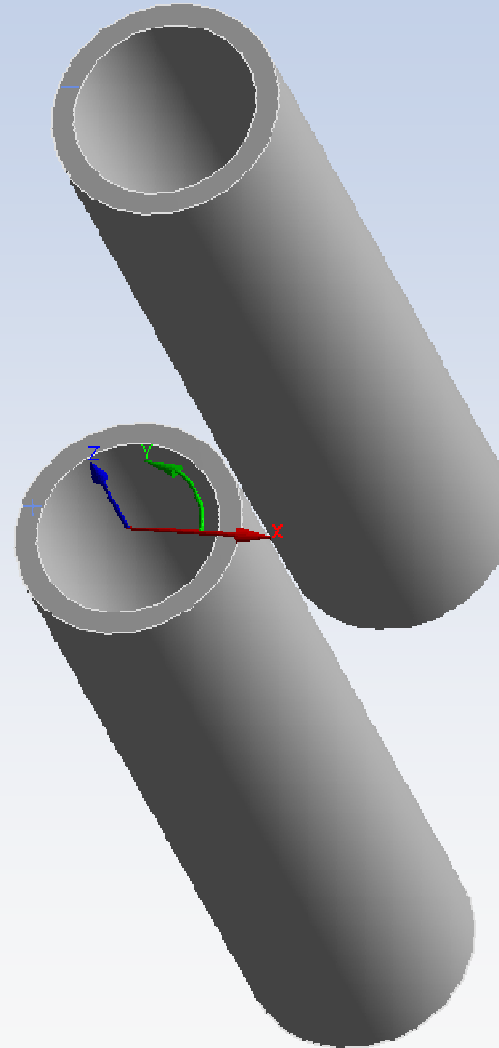




# Directional dependency



- **Products built from certain materials, have directional properties.**
- **By using coordinate systems and orthotropic properties we can capture these effects**
- **Straightforward for cylinders, plates etc.**





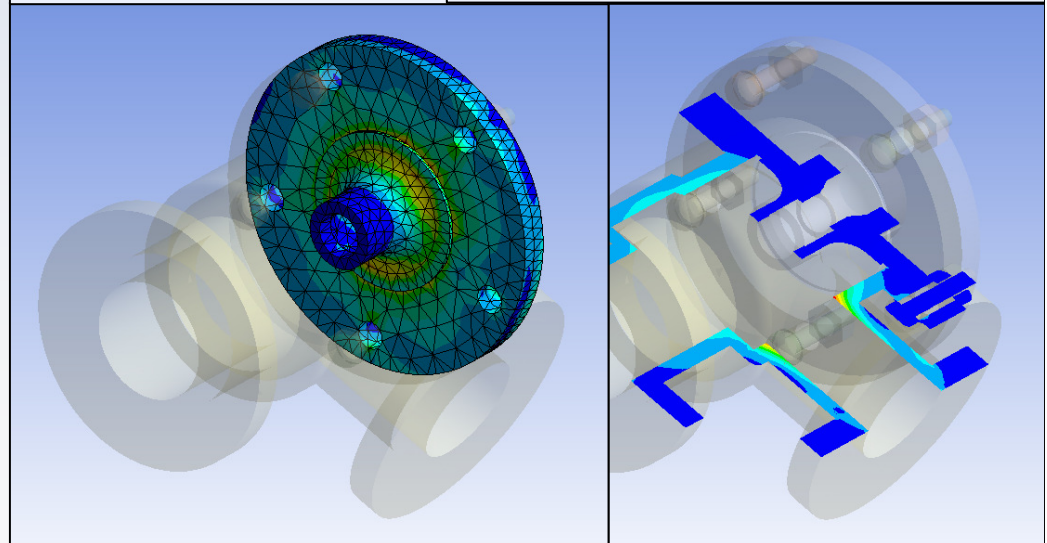
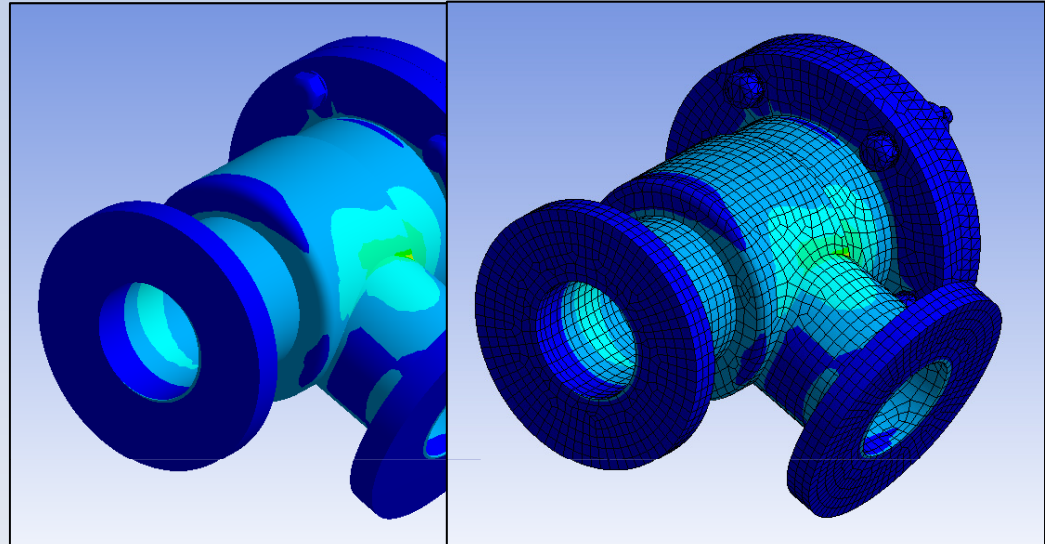
# ANSYS Post Processing



# Results plots

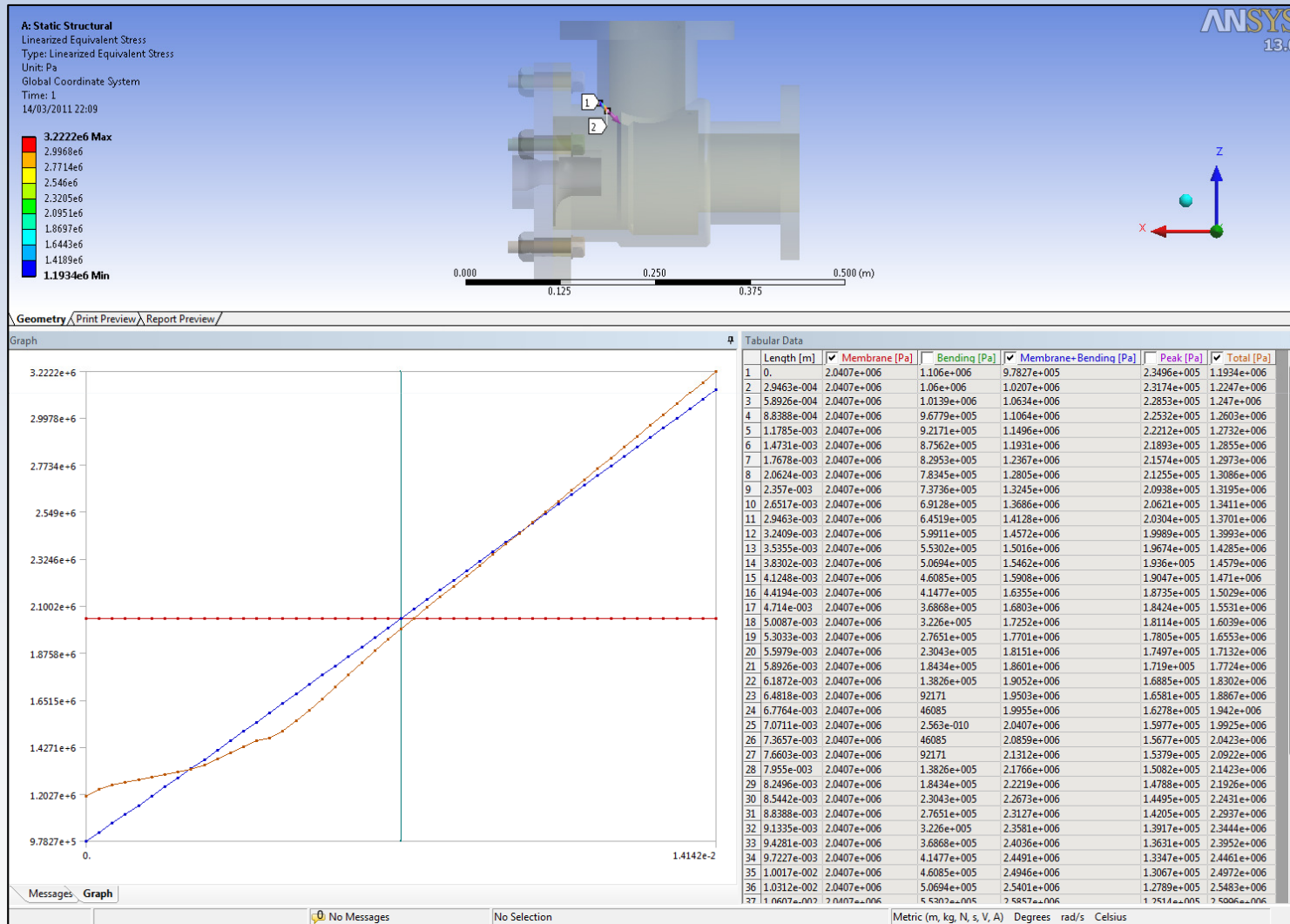


- Whole assembly
- With mesh
- Scoped results
- Cut planes
- Parametric feedback



[-] Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
[-] Definition	
Type	Equivalent (von-Mises) Stress
By	Time
Display Time	Last
Calculate Time History	Yes
Identifier	
[-] Integration Point Results	
Display Option	Averaged
[-] Results	
<input type="checkbox"/> Minimum	2059.1 Pa
<input type="checkbox"/> Maximum	4.6807e+005 Pa
[+] Information	

# Results - linearisation



# Reports



- Automatic reporting
  - Captures all CAE data
  - HTML
  - Word
  - PowerPoint





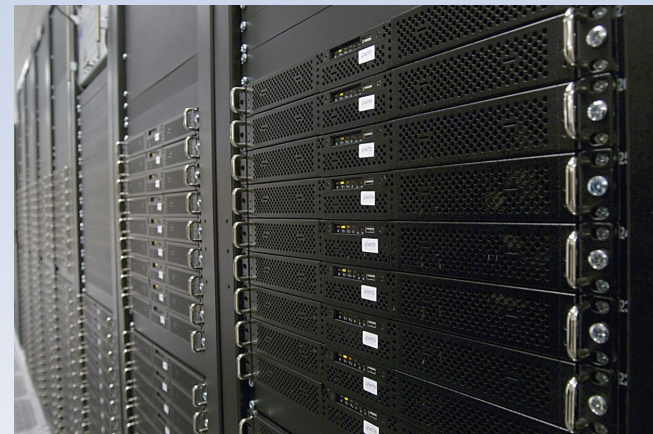
# ANSYS Optimisation



- **Start with a workbench defined process**

- Use parameters in:

- CAD
- DesignModeler
- SpaceClaim
- Engineering data
- Preprocessor
- Postprocessor
- Derived parameters



- Define design envelope

- Parameter limits
- Discrete/continuous

- Define goal

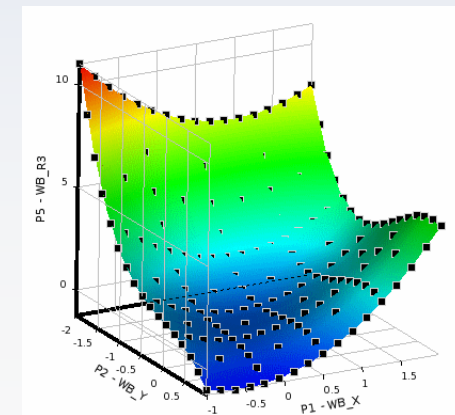
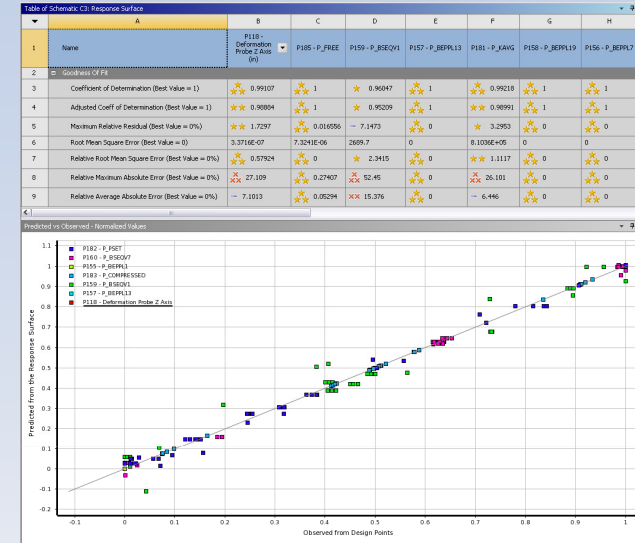
- Near target
- Min/Max
- Trade offs

Design of Experiments	
Design of Experiments Type	Central Composite Design
Design Type	Central Composite Design
	Optimal Space-Filling Design
	Box-Behnken Design
	Custom
	Custom + Sampling
	Sparse Grid Initialization

Table of Design Points						
	A	B	C	D	E	F
1	Name	P1 - InletLen...	P2 - PlateTh...	P3 - Plenum..	P4 - InletDia...	P5 - Vinlet
2						m s^-1
3	Current	15	0.5	10	7	1
4	DP 1	15	0.2	10	7	2
5	DP 2	15	0.1	10	7	3
6	DP 3	20	0.5	10	6	4
7	DP 4	20	0.5	10	6	5

- **Six Sigma analysis**

- Manufacturing tolerances
- Perfect design becomes real world design





# Demonstration



- **Geometry**
- **Meshing**
- **Preprocessing**
- **Solve**
- **Post**



Thank you



# The Days Agenda



10:00 – 10:30	<b>Registration</b>
10:30 – 10:45	<b>Intro to Seminar &amp; ANSYS</b>
10.45 – 11.45	<b>Structural Mechanics Analysis using ANSYS Workbench</b> <ul style="list-style-type: none"><li>• Overview of Workbench for Structural Analysis</li><li>• CAD Connectivity</li><li>• Geometry Pre-Processing</li><li>• Meshing</li><li>• Materials Support</li><li>• Boundary Condition Setup</li><li>• Analysis Types</li><li>• Post Processing</li><li>• Optimisation</li></ul>
11.45 – 12.15	<b>Coffee Break</b>
12.15 – 13.15	<b>ANSYS Composite modelling</b> Presented by Even Evolutionary Engineering AG
13.15 – 13.30	<b>Q &amp; A Sessions</b>
13:30 – 14:30	<b>Lunch &amp; Finish</b>