



real inspiration

SimulationXpress

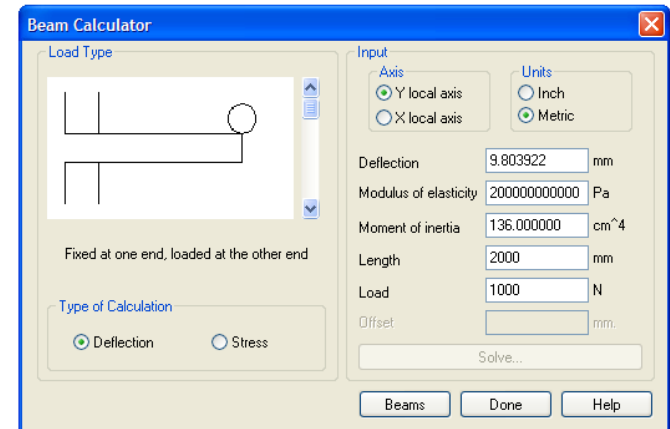
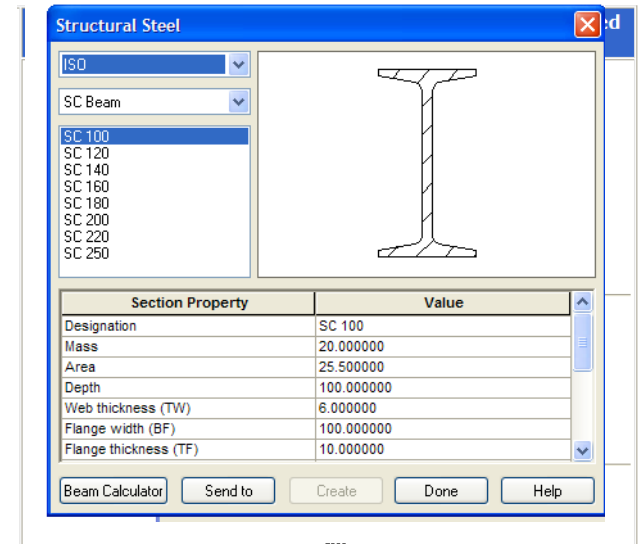
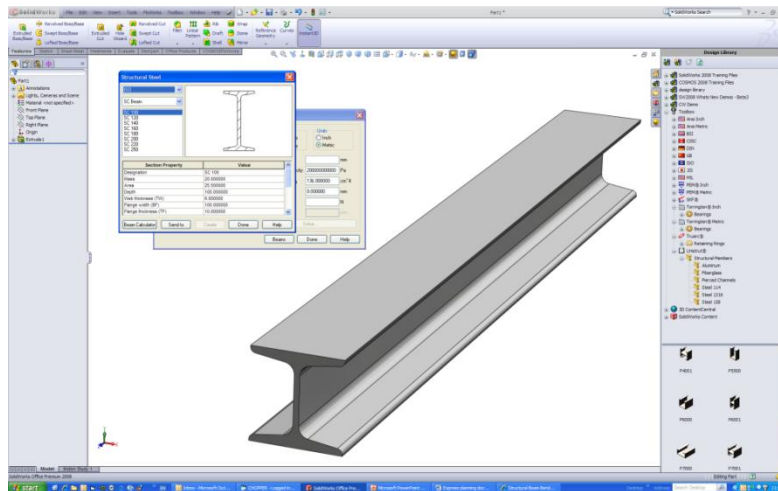
Agenda

- Traditional stress calculations
- What is Finite Element Analysis (FEA)?
- How do I use SimulationXpress?
- Case Study 1 - Idler Arm
- Exercises
 - Pump Cover
 - Ratchet Body
- Results
 - How do I interpret the results?
 - How do I know if results are accurate?
- Convergence group exercise
- Solving and Optimising
 - Case Study 2 - Cast Bracket Optimisation
- Optimisation exercises
 - Casting
 - Pulley
- Assumptions in SimulationXpress
- Getting results out
- Tips and Tricks
- Going further with FEA



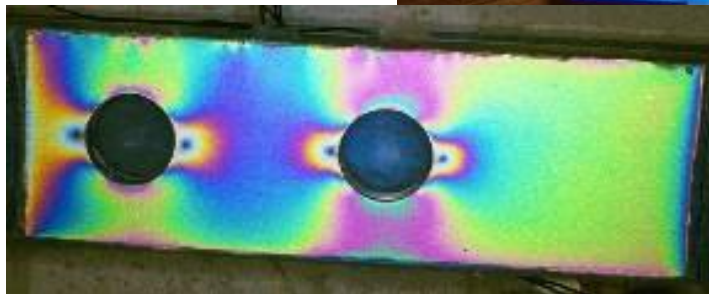
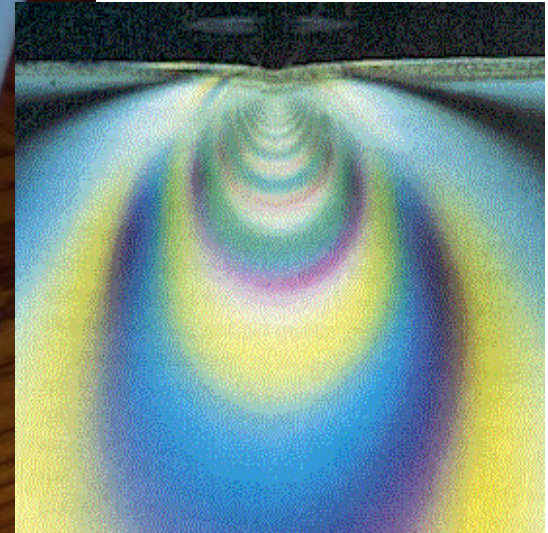
Traditional Stress Calculations

- Mathematical formulae
 - Stress
 - Deflection
- Limited to certain simple shapes
- Could use 'Beam Calculator'



Traditional Stress Evaluation

- Prototypes
- Scale models
- Photoelasticity



Expensive

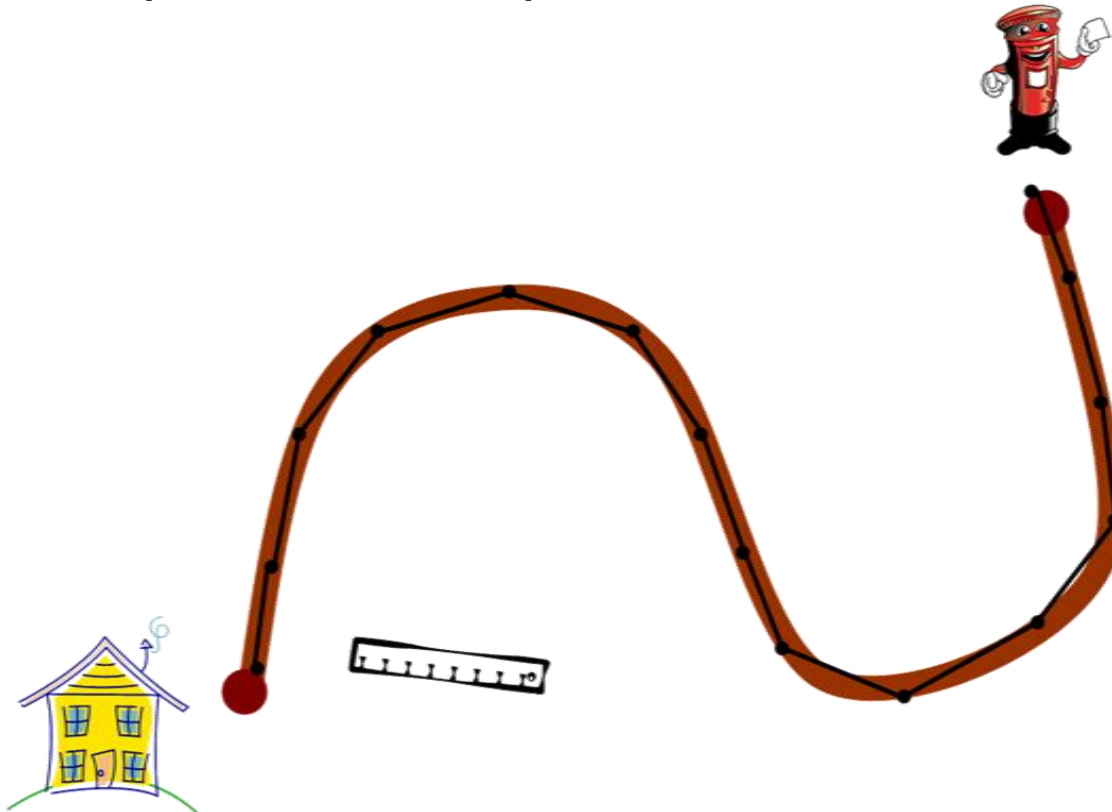
Time Consuming

Limited Information



What is Finite Element Analysis?

- Example: How to measure a curved path from home to a post box
 - Curved path is approximated/simplified by straight segments (such as metres)



What is Finite Element Analysis?

- Instead of a metre ruler, we use an Element

Stiffness equation:

$$F = K \times D$$

(Force = Stiffness x Deflection)

$$F1 = K1 \times D1$$

$$F2 = K2 \times D2$$

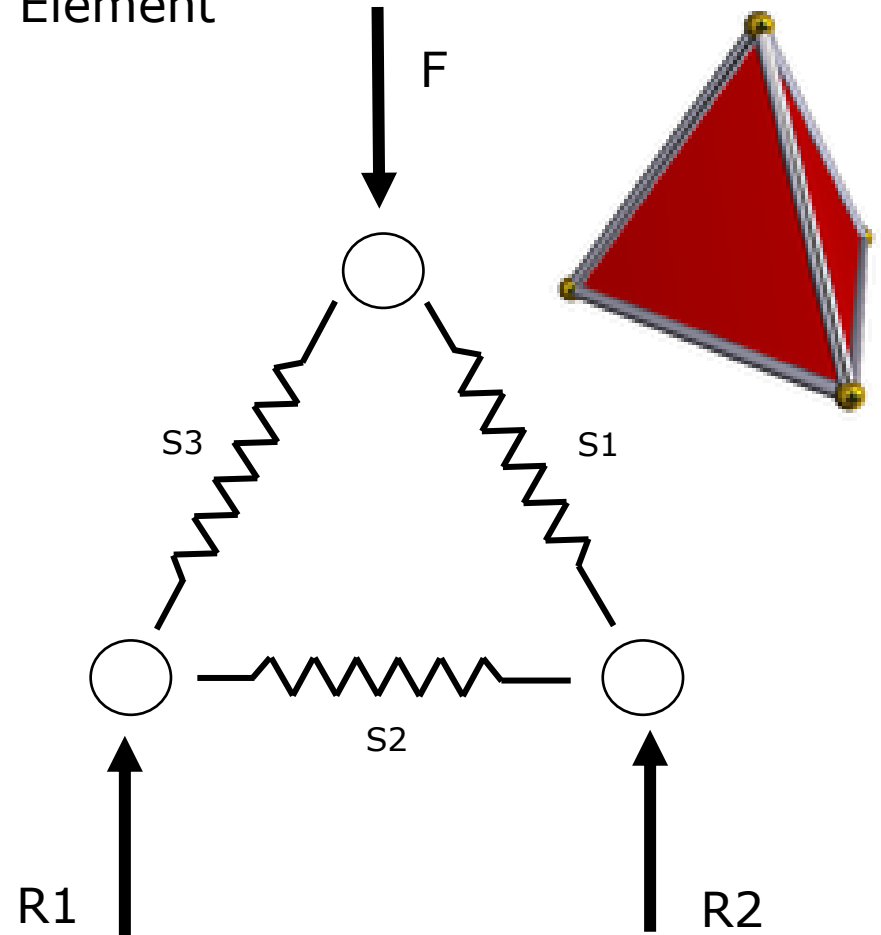
$$F3 = K3 \times D3$$

(Stiffness Matrix)

Equilibrium equations:

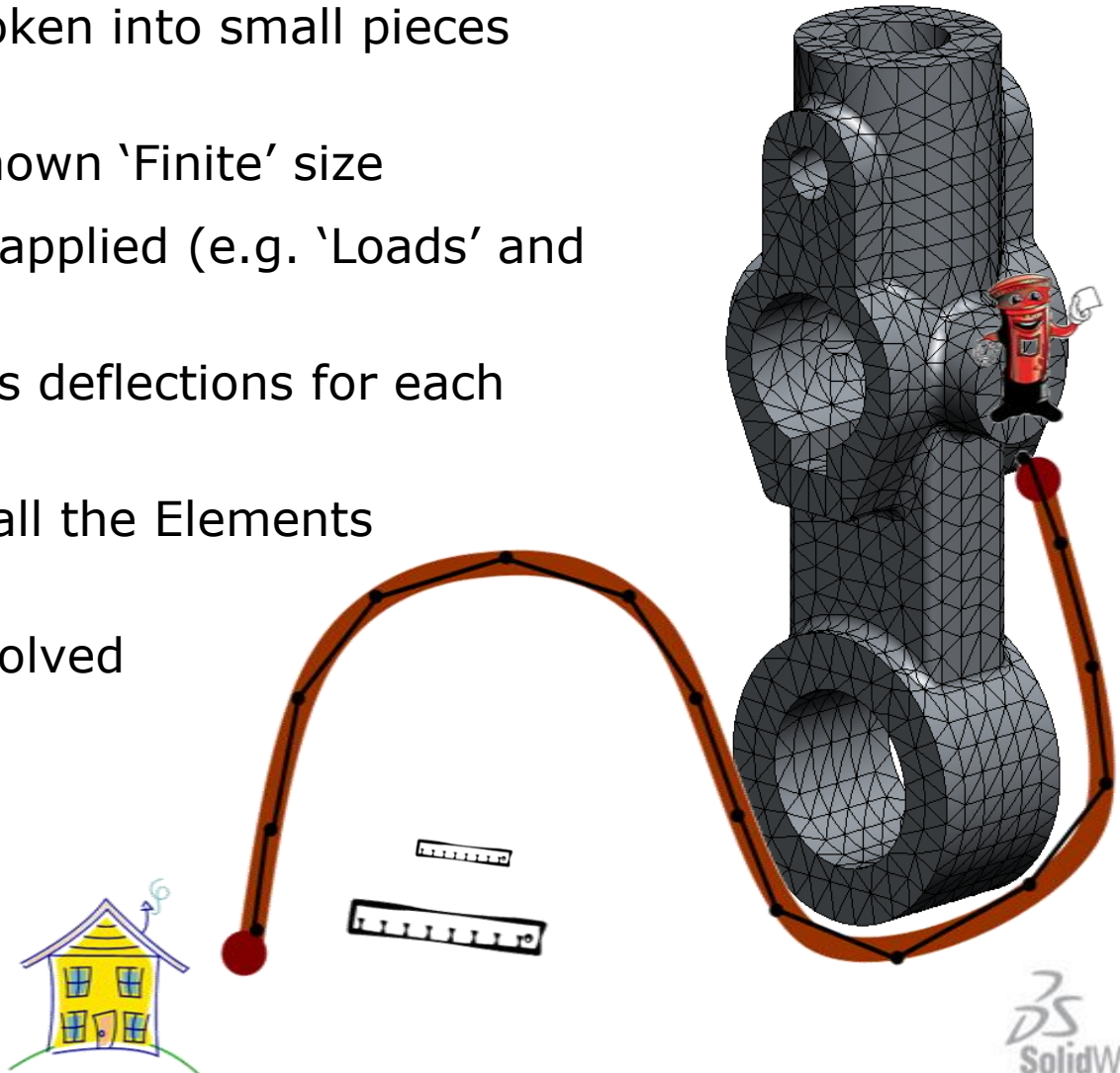
$$F = R1 + R2$$

$$D1 = D2 = D3$$



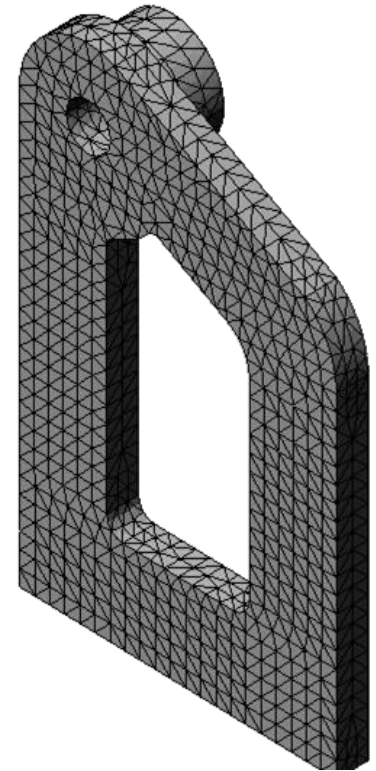
What is Finite Element Analysis?

- Complex geometry broken into small pieces ('Elements')
- Each Element has a known 'Finite' size
- Overall conditions are applied (e.g. 'Loads' and 'Restraints')
- The software calculates deflections for each Element
- Equations created for all the Elements automatically
- All the equations are solved



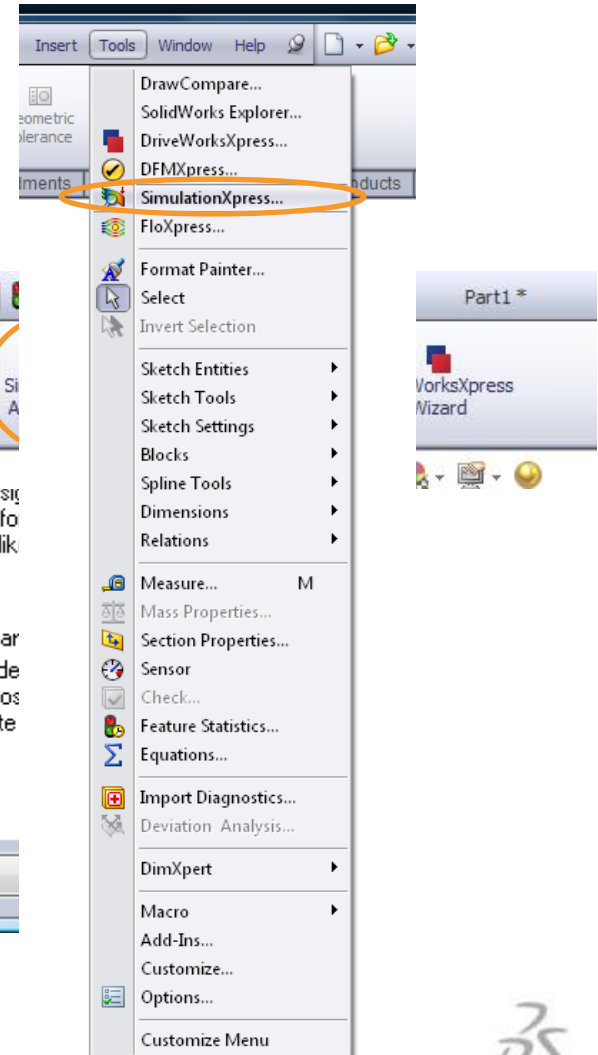
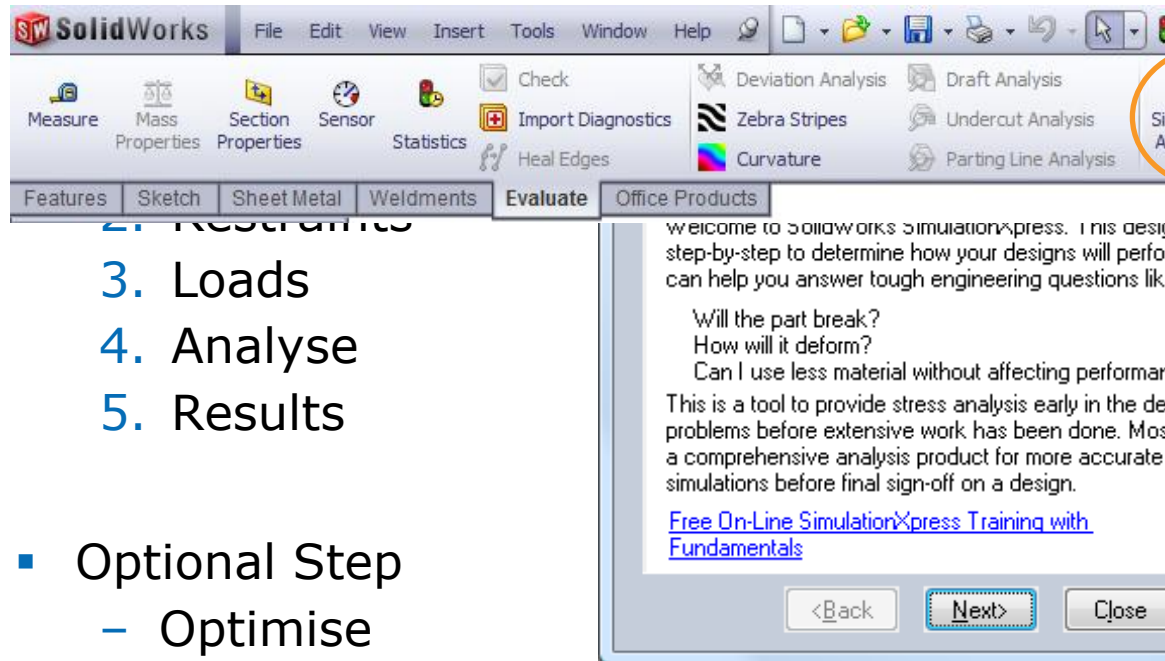
Definitions: Elements, Mesh, Nodes

- Elements
 - In SimulationXpress - Tetrahedrons
 - Other Elements available in Simulation
- Mesh
 - The sum of all the Elements
 - Approximate representation of the CAD geometry using the Tetrahedral Elements
- Nodes
 - Points at which different Elements are joined together; locations where values of unknowns (usually displacements) are to be approximated



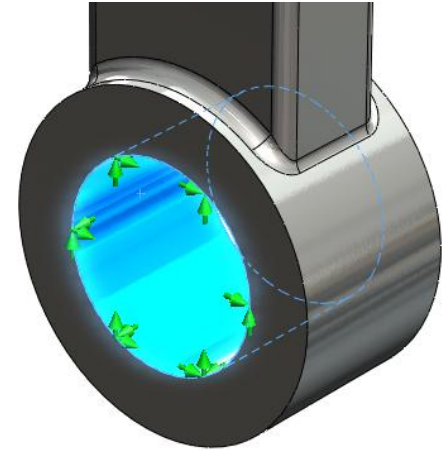
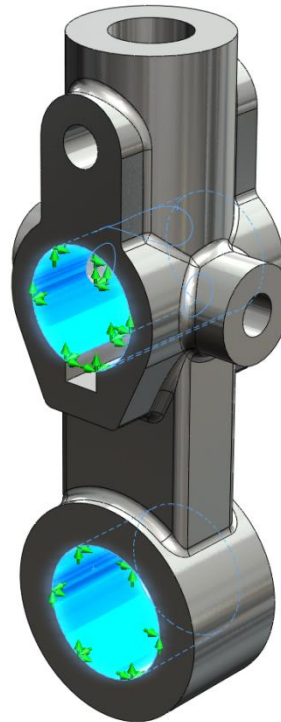
How Do I Use SimulationXpress?

- Where is it?
 - Command Manager
 - Tools > SimulationXpress



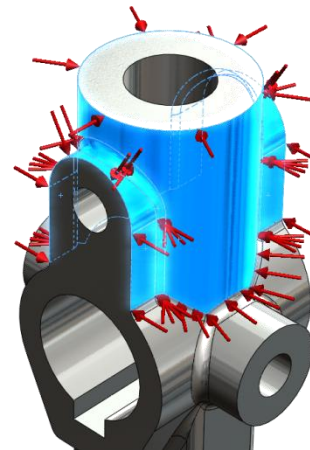
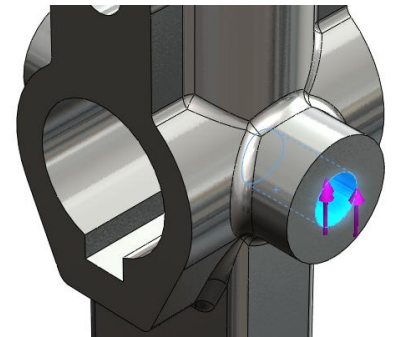
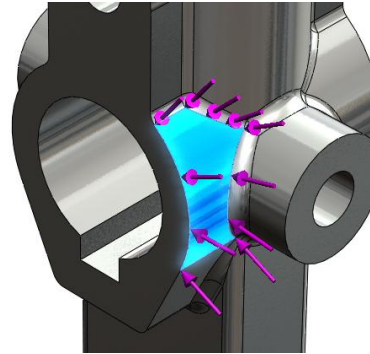
Using the Wizard - Restraints

- Fixed Restraint
 - All 6 degrees of freedom are removed
 - Translation in X, Y, Z
 - Rotation about X, Y, Z
- Multiple Restraints



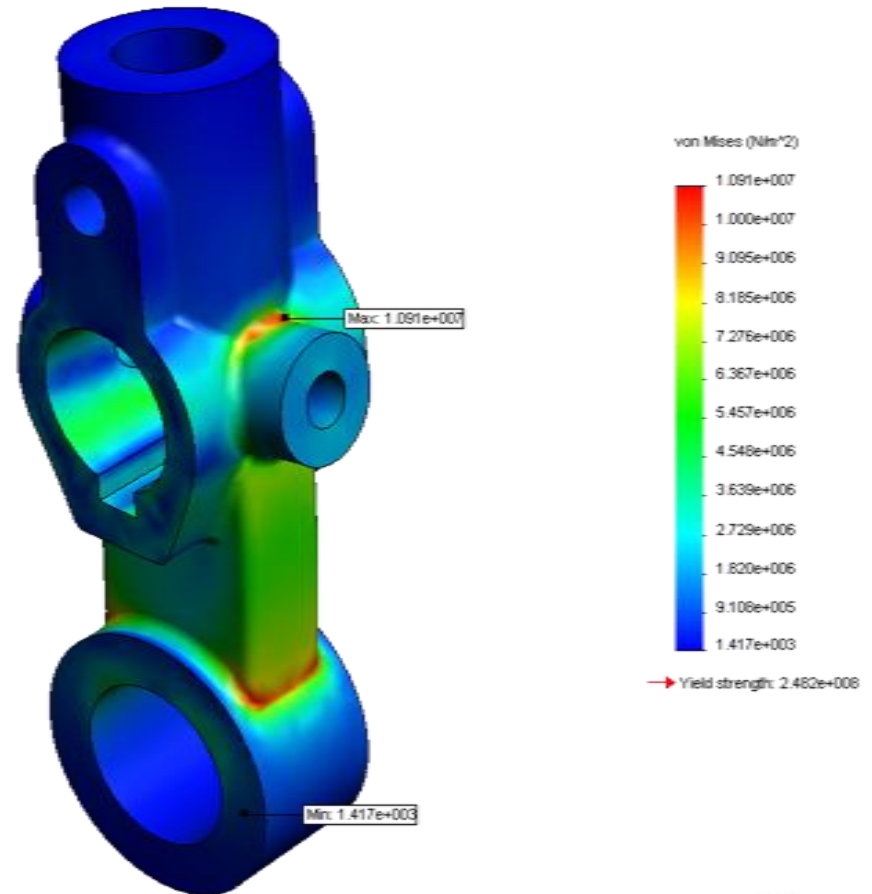
Using the Wizard - Loads

- Forces
 - Applied to one or more faces
 - Direction required:
 - Normal to face
 - Normal to Plane
- Pressures
 - Applied to a face
 - No direction required
- Can have multiple forces or pressures



Case Study 1 - Idler Arm

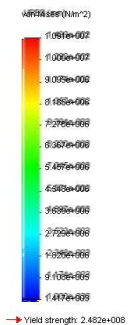
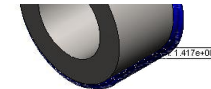
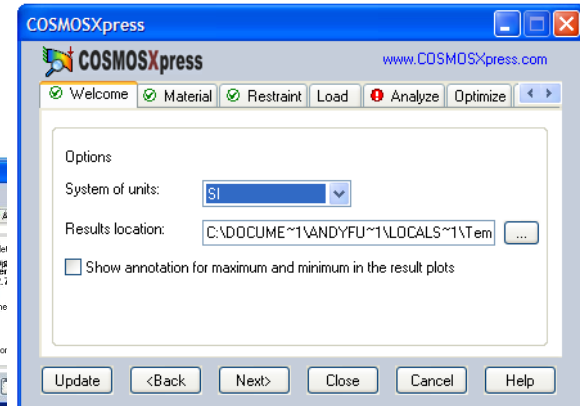
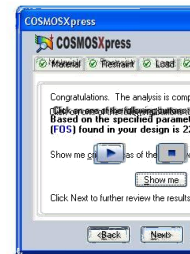
Model name: Idler Arm
Study name: COSMOSXpressStudy
Plot type: Static nodal stress Plot1
Deformation scale: 1417.53



How do I interpret results?

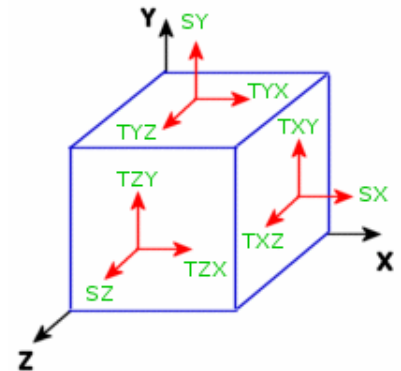
- 4 results plots
 - FOS
 - Von Mises Stress
 - Displacement
 - Deformed Shape
- Note scale!
- Can choose units in 'Options'
 - SI or English
- Can switch on 'Maximum' and 'Minimum' flags in plots

Model Results - FOS
Show Results - COSMOSXpress Results
FOS (FOS) - FOS (FOS) - FOS (FOS)
Red < FOS < 30 < Blue



Using stress results to validate design

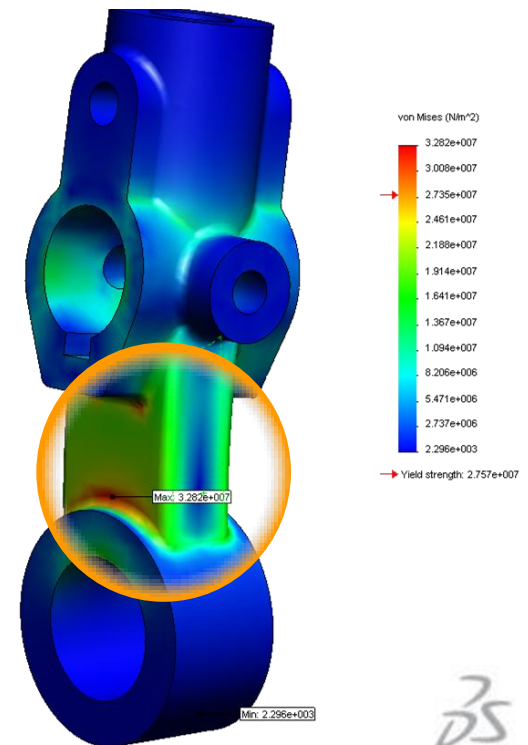
- FOS & von Mises Stress, how do these relate together and how can we use them to validate our design?
- **Factor of Safety** =
$$\frac{\text{Limit stress for the material}}{\text{Maximum **von Mises** stress in the model}}$$
- Limit Stress for the Material
 - Simple uniaxial loading information based on test data
 - Contained within the material properties from SW database
- Maximum von Mises Stress (Richard Edler von Mises)
 - 3D State of Stress
 - 9 stress components overall
 - (3 direct stresses and 6 shear stresses)
 - Von Mises stress is independent of coordinate system



- von Mises stress =
$$\sqrt{0.5[(SX - SY)^2 + (SY - SZ)^2 + (SZ - SX)^2] + 3(TXY^2 + TYZ^2 + TZX^2)}$$

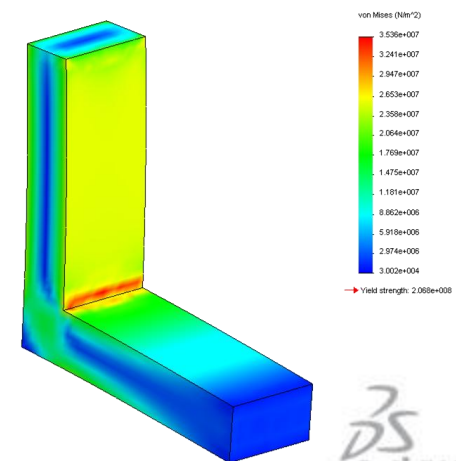
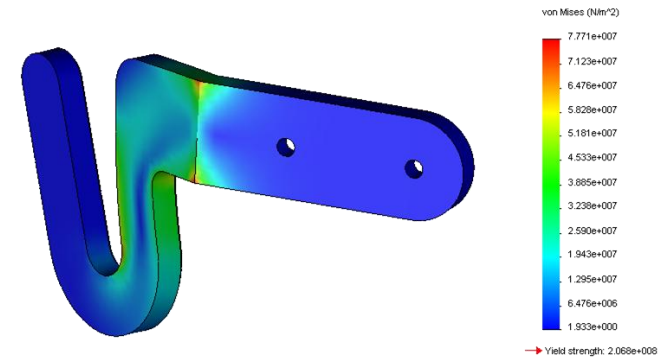
How do I interpret results?

- Interpreting maximum stress
- If it has a region above von Mises yield stress, will it break?
 - Maybe
 - Probably not!
- Can it redistribute the Load elsewhere?



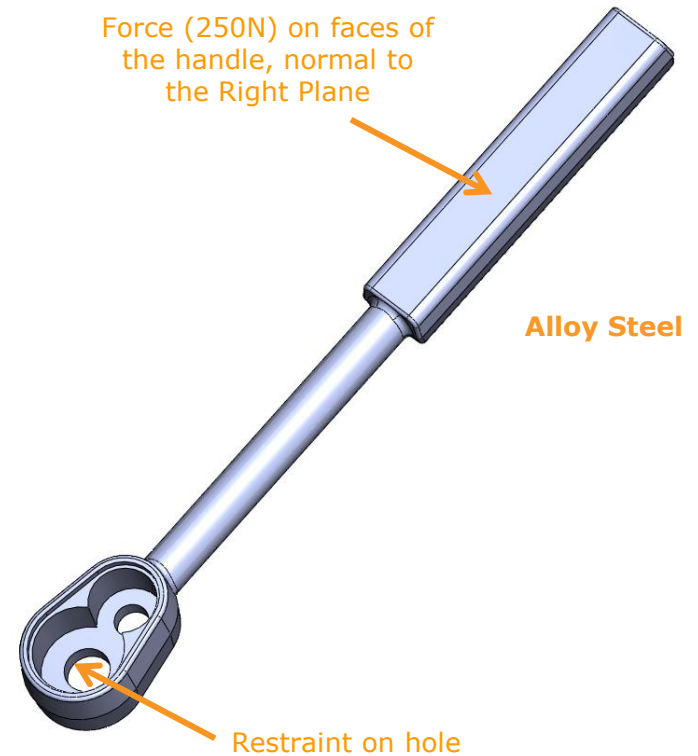
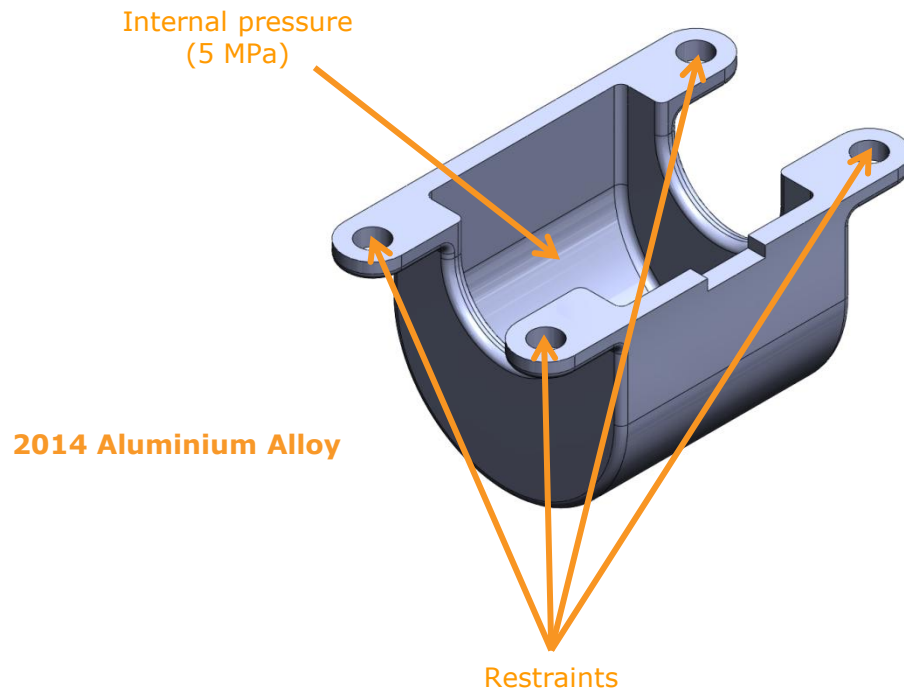
How do I know if results are accurate?

- Accuracy depends on:
 - How it is Restrained and Loaded
 - Size of Elements
 - Singularities (maths issues)
- In SimulationXpress
 - Regions around Restraints and Loads may show high peak stresses
- Convergence
 - Can be used to study the effects of Mesh size
- Singularities
 - No part has perfectly zero radius corners



Exercises (‘SimulationXpress Training Seminar’ folder on Desktop)

- Pump Cover
- Ratchet Body



Convergence – group exercise

- Plate with hole - under 10kN tension
- What are the effects of varying Element size?
- How do I vary Element size?

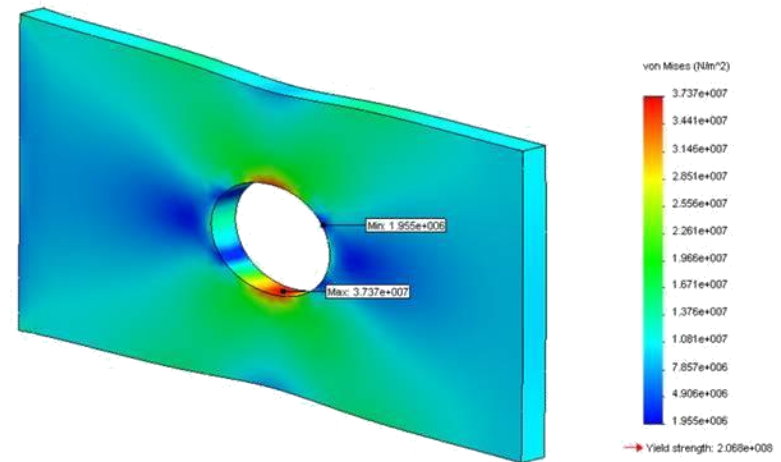
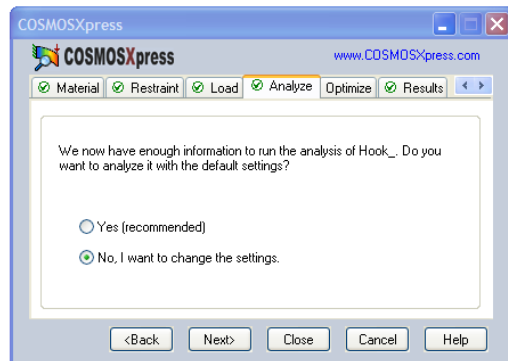
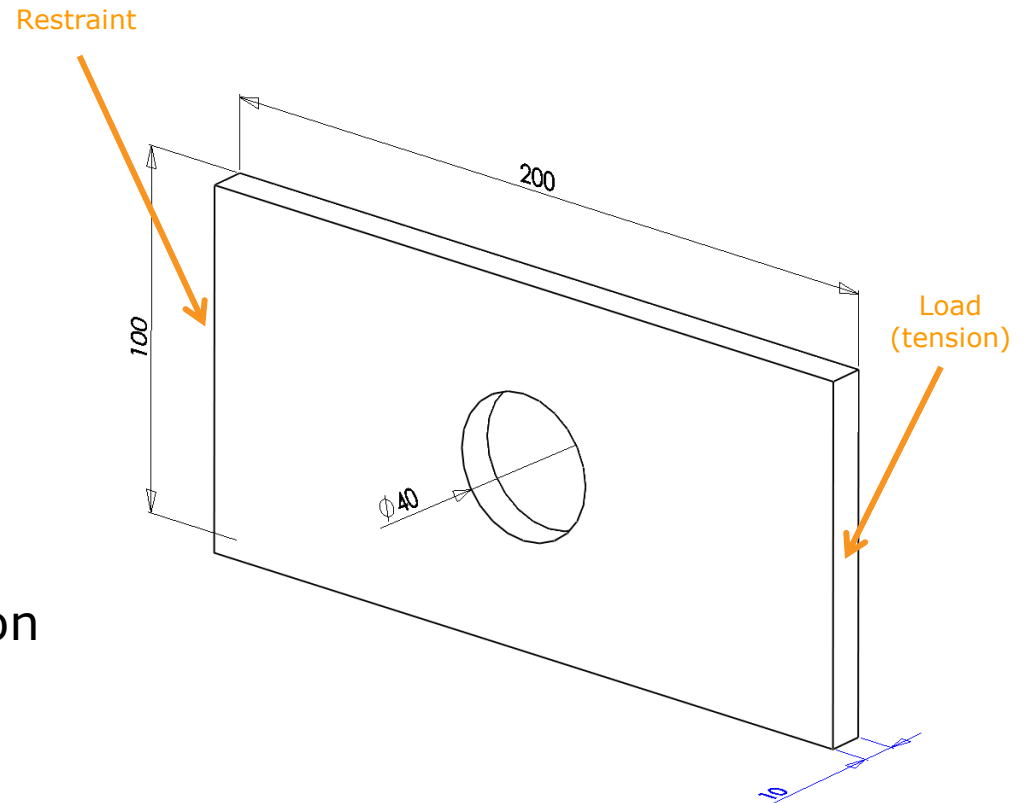


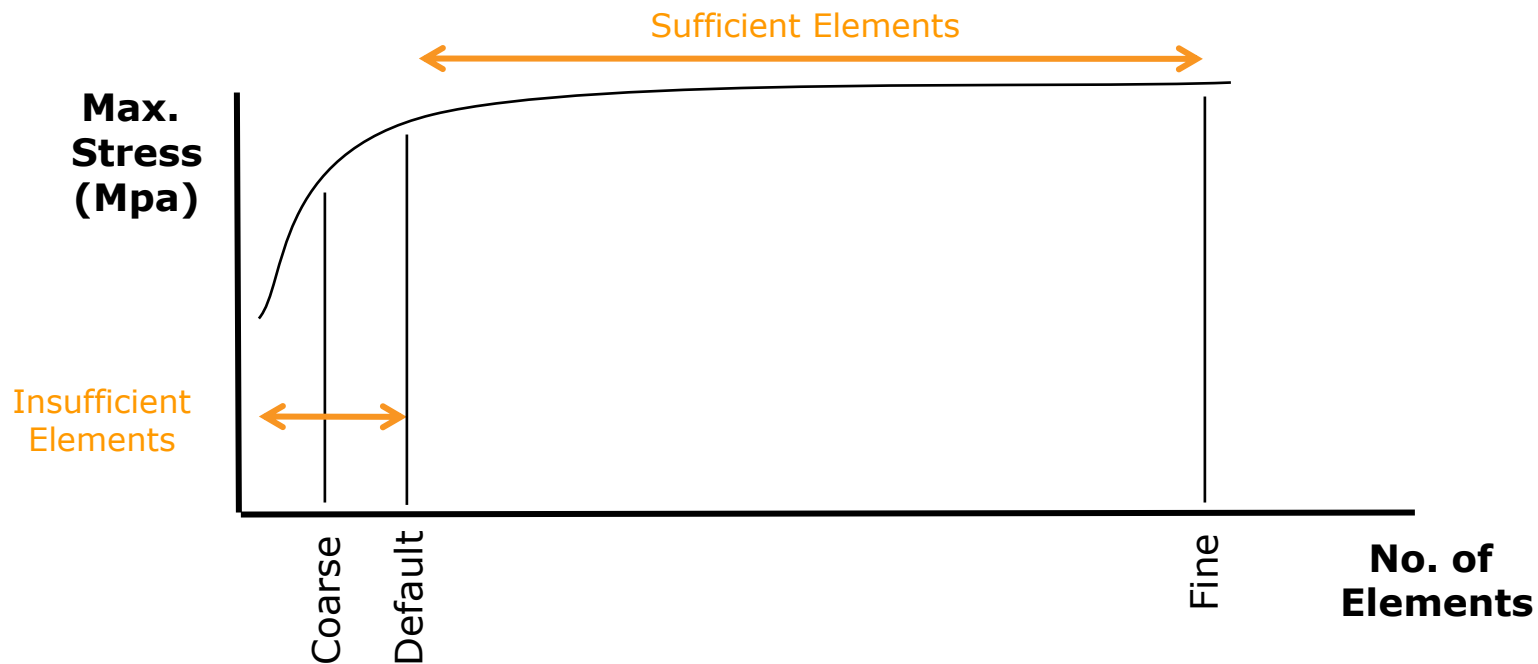
Plate details

- Length = 200 mm
- Height = 100 mm
- Width = 10 mm
- Hole = Dia 40 mm
- Hole is located centrally
- Material = AISI 304 (Steel)
- Load (tension) = 10kN force
- Make sure max. and min. annotations are turned on



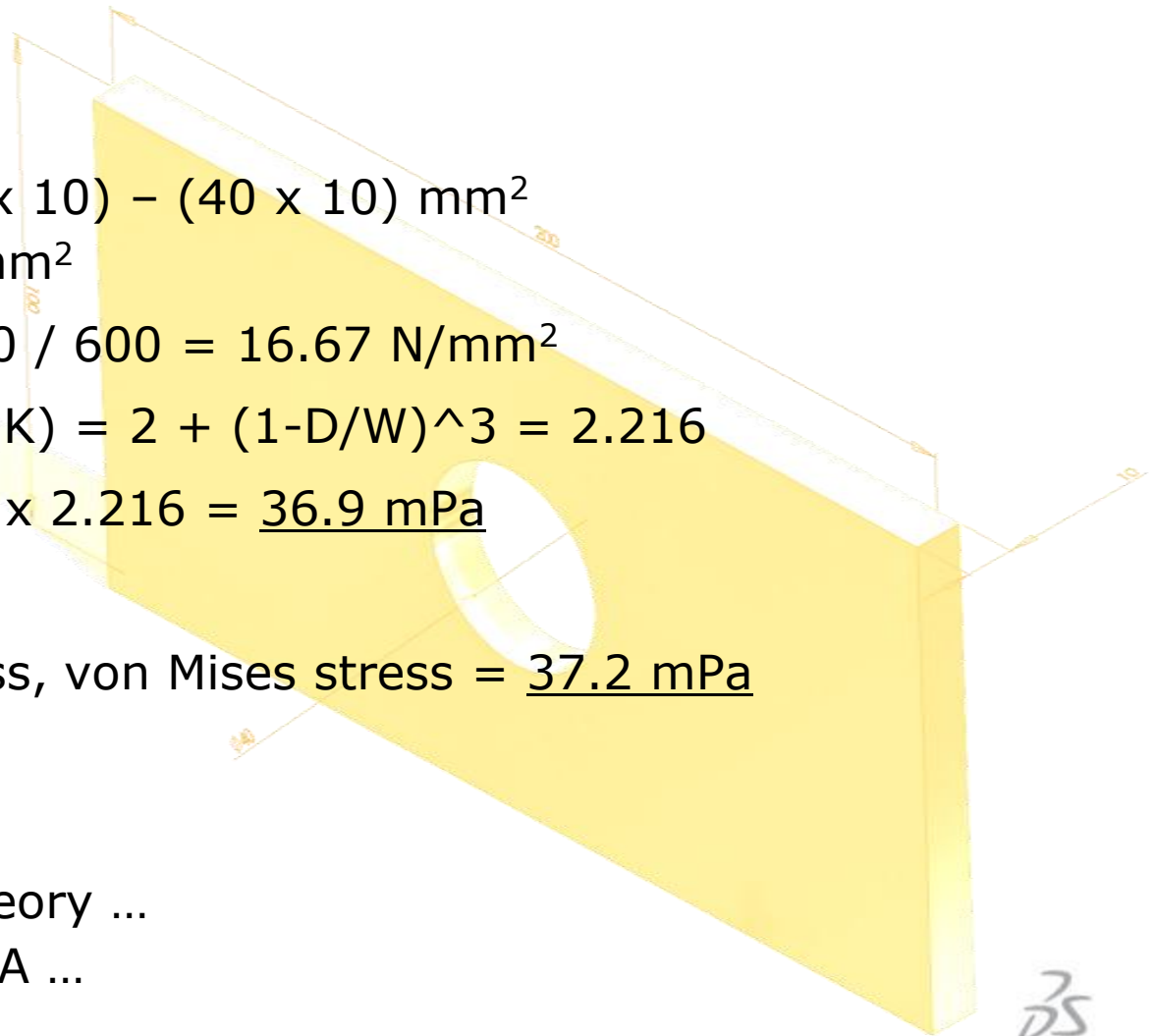
Convergence results

	Coarse Mesh	Standard Mesh	Fine Mesh
Element size (mm)	11.45	5.72	2.86
No. of Elements	1,373	7,017	55,321
Max Stress (Pa)	3.54×10^7	3.72×10^7	3.78×10^7
Max Displacement (mm)	1.30×10^{-2}	1.30×10^{-2}	1.31×10^{-2}



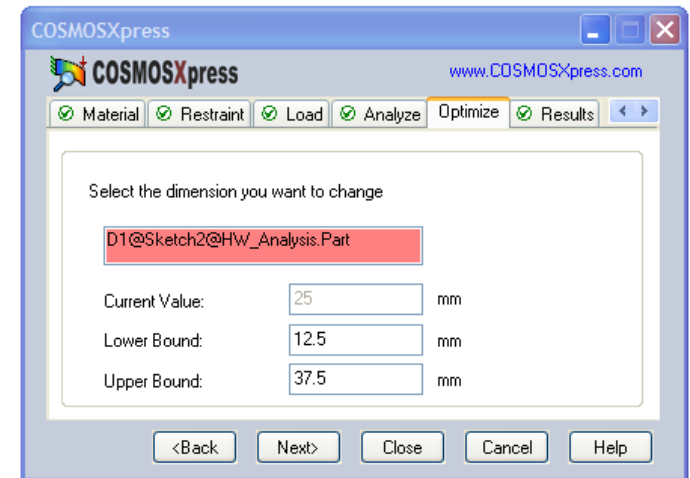
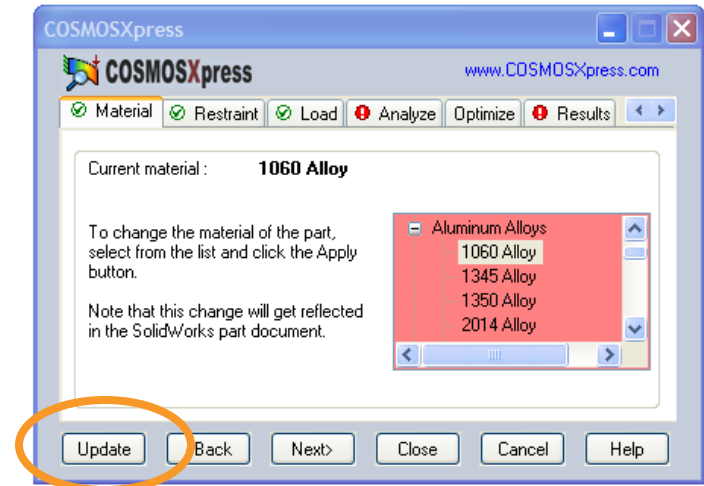
Comparison with theory

- Stress = Force / Area
- Force = 10,000 N
- Area @ hole = $(100 \times 10) - (40 \times 10) \text{ mm}^2$
= 600 mm²
- Direct stress = $10,000 / 600 = 16.67 \text{ N/mm}^2$
- Concentration factor (K) = $2 + (1-D/W)^3 = 2.216$
- Direct stress = $16.67 \times 2.216 = \underline{36.9 \text{ mPa}}$
- From SimulationXpress, von Mises stress = 37.2 mPa
- Why difference?
 - Assumptions in theory ...
 - Assumptions in FEA ...



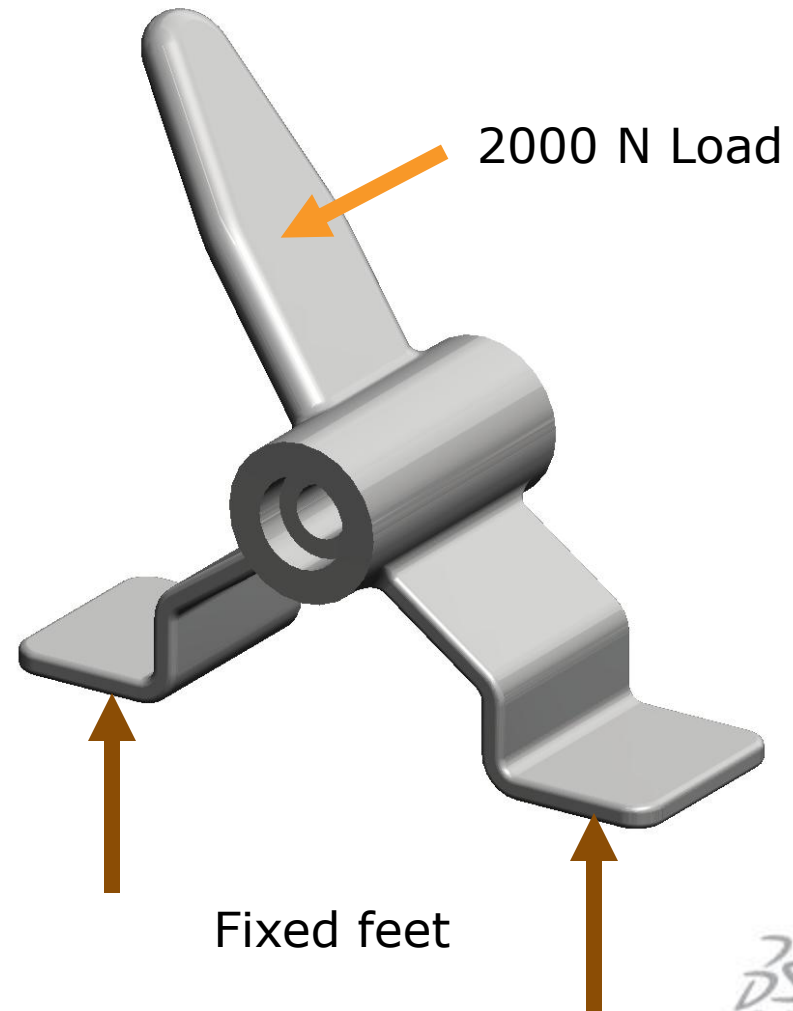
Solving and Optimising

- Solver
 - 'Run' to solve initially
 - 'Update' button after changes
- Optimisation
 - Optional
 - Choose a criteria to Optimise on
 - Min. FOS
 - Max. Stress
 - Max. Deflection
 - Select 1 dimension to vary
 - Add limits
 - Can do repeated Optimisation runs



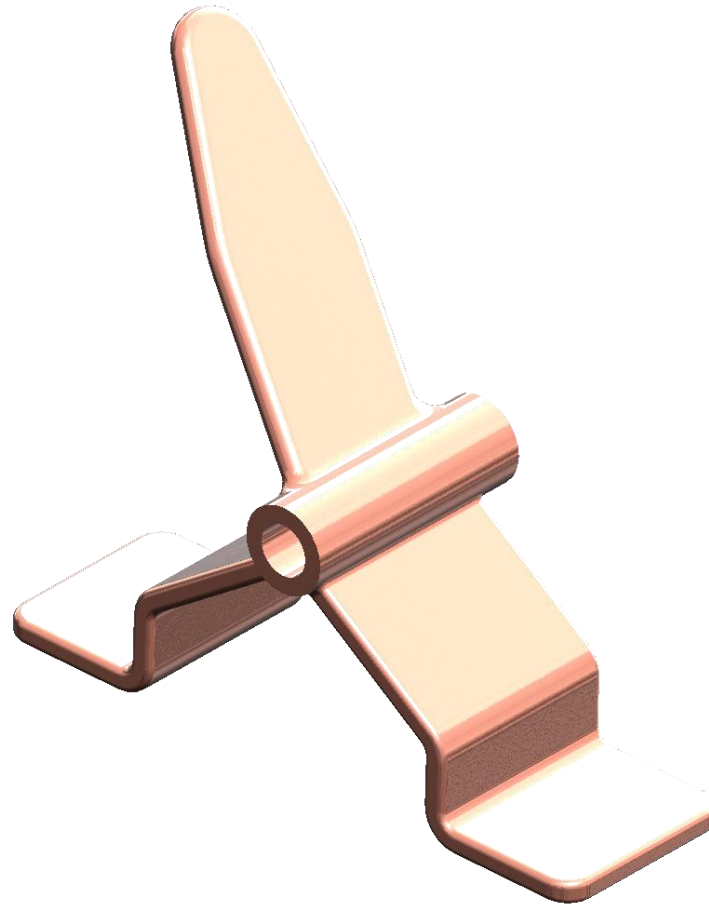
Case Study 2 - Cast Bracket Optimisation

- Design requirements:
 - $FOS > 2$
 - Minimise material
- Material
 - Cast Carbon Steel
- Working Load
 - 2000 N force on blade
- Fixing
 - As if welded to a flat surface



Case Study 2 - Cast Bracket Optimisation

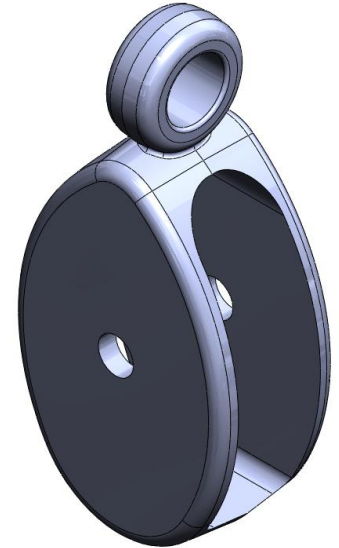
- Initial Mass
 - 6.47 kg
- After Optimise 1
 - 5.89 kg
- After Optimise 2
 - 4.01 kg
- Saving of 38% mass overall



Optimisation exercises (‘SimulationXpress Seminar’ folder on Desktop)

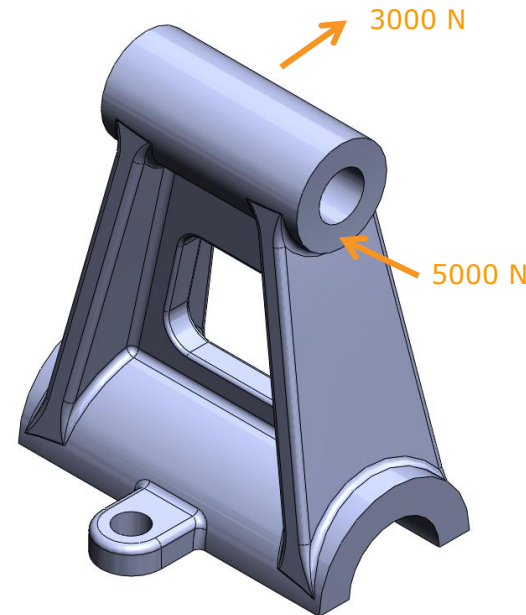
■ Pulley

- Material: Aluminium 1060
- Objectives
 - $FOS > 3$
- Restrain eye
- Apply 10N loads on each hole vertically downwards
- Optimise the weak point!



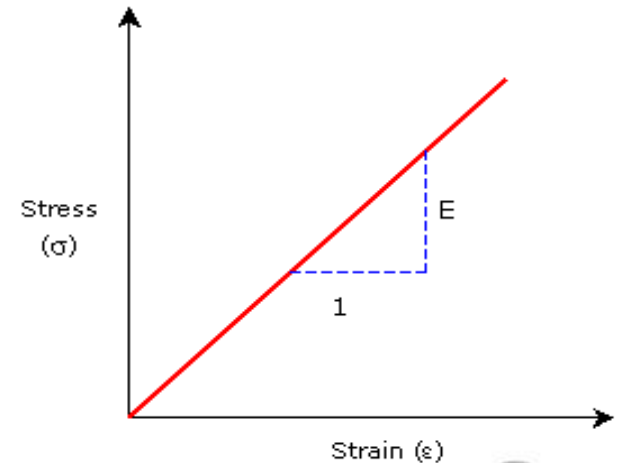
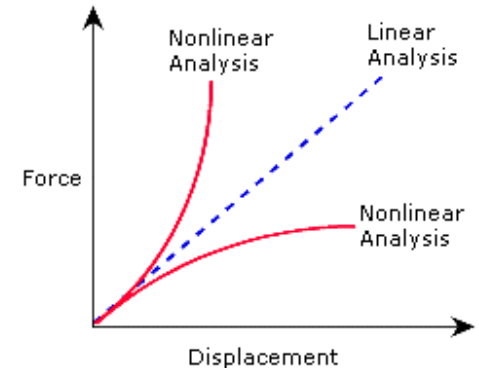
■ Casting

- Material: Cast Alloy Steel
- Objective
 - Minimise mass and
 - $VM\ Stress < 150\ MPa$
- Restrain bottom
- 2 loads
 - 3 kN in hole normal to front
 - 5 kN on end face of upper boss
- Optimise using the 8 mm dim
- Note the use of link values
- What % saving in mass is possible?



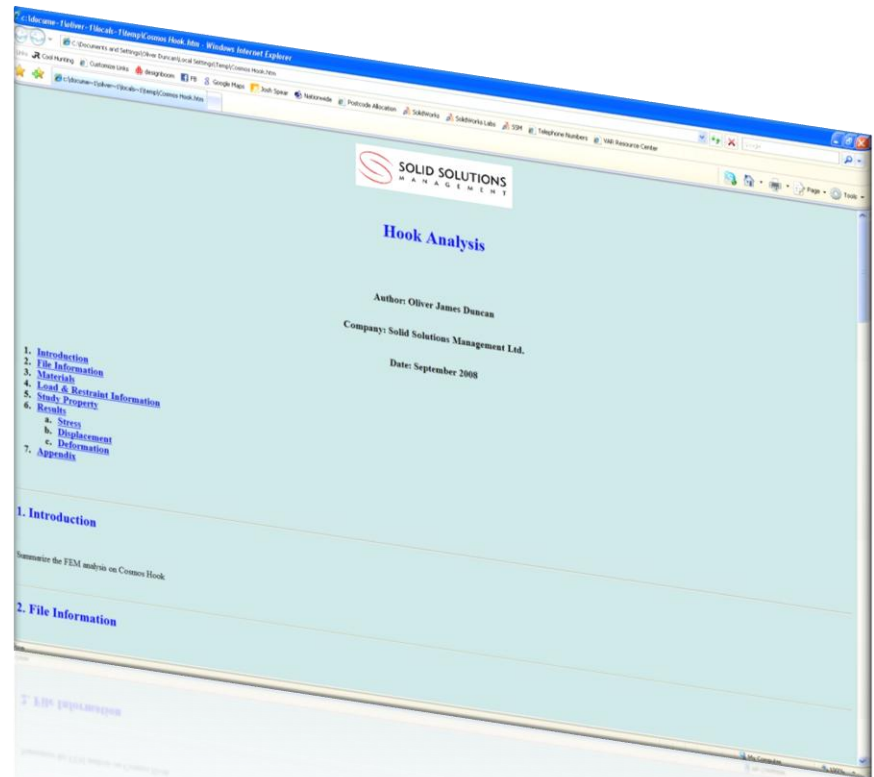
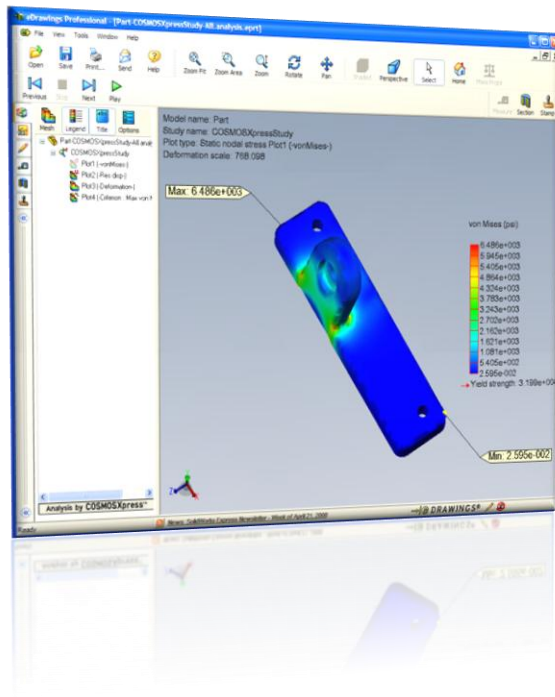
Assumptions in SimulationXpress

- Material is linearly elastic
 - If you double the load, deformation also gets doubled
 - The part returns to its original shape if the loads are removed (no permanent deformation)
- Loads are static
 - Loads are applied slowly and gradually
 - Rapidly-applied loads cause additional displacements, strains, and stresses
- Small displacement (less than 5%)
 - Stiffness change when material is deformed is not taken into account



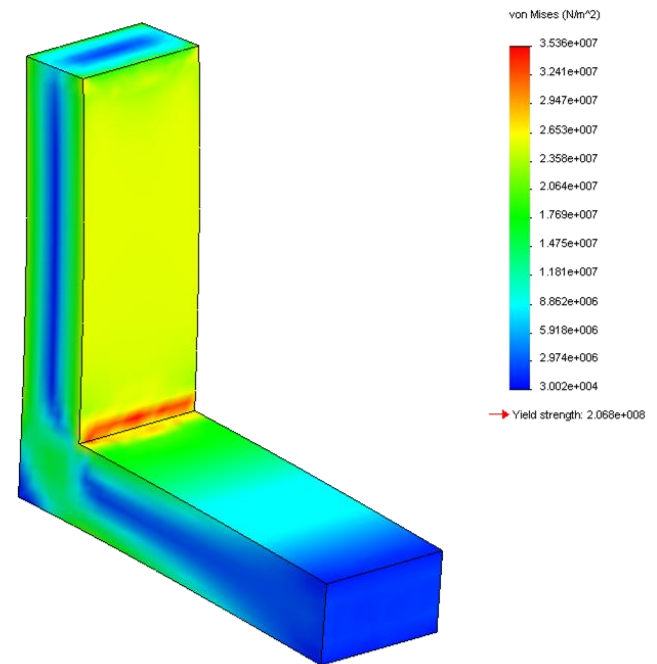
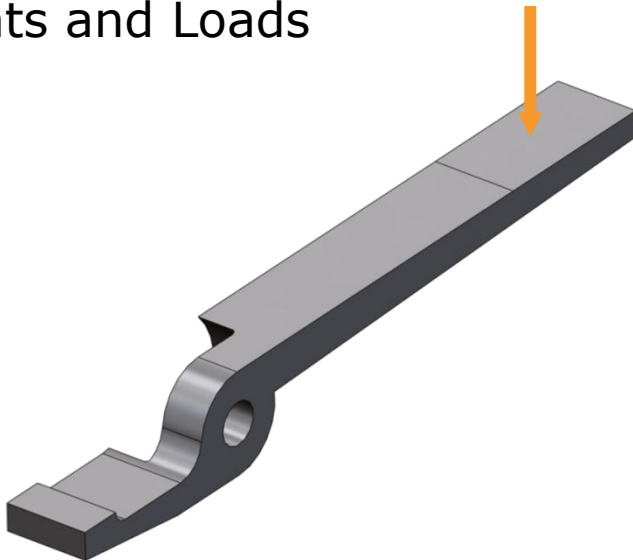
Getting results out

- Report Writer
- Animations
- Print Screen
- eDrawings



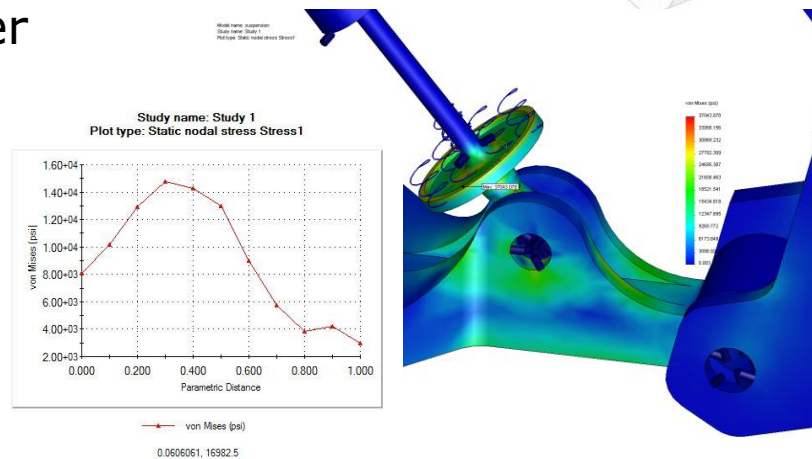
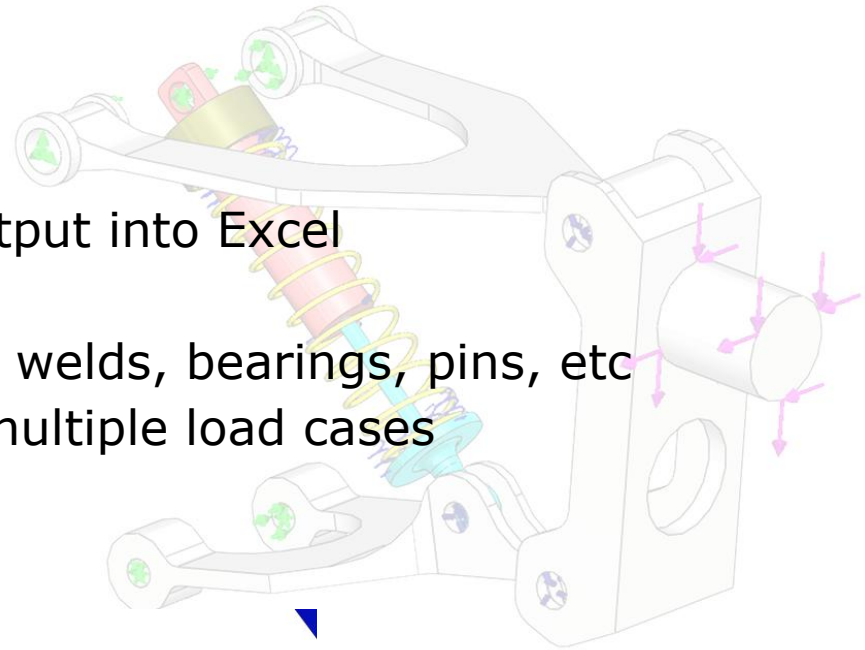
Tips and Tricks

- Orientate the part prior to report creation
- Problems with thin parts and weldments
- Force – per entity!
- Rename your Restraints and Loads
- Understand singularities
- Use split lines for more accurate Restraints and Loads



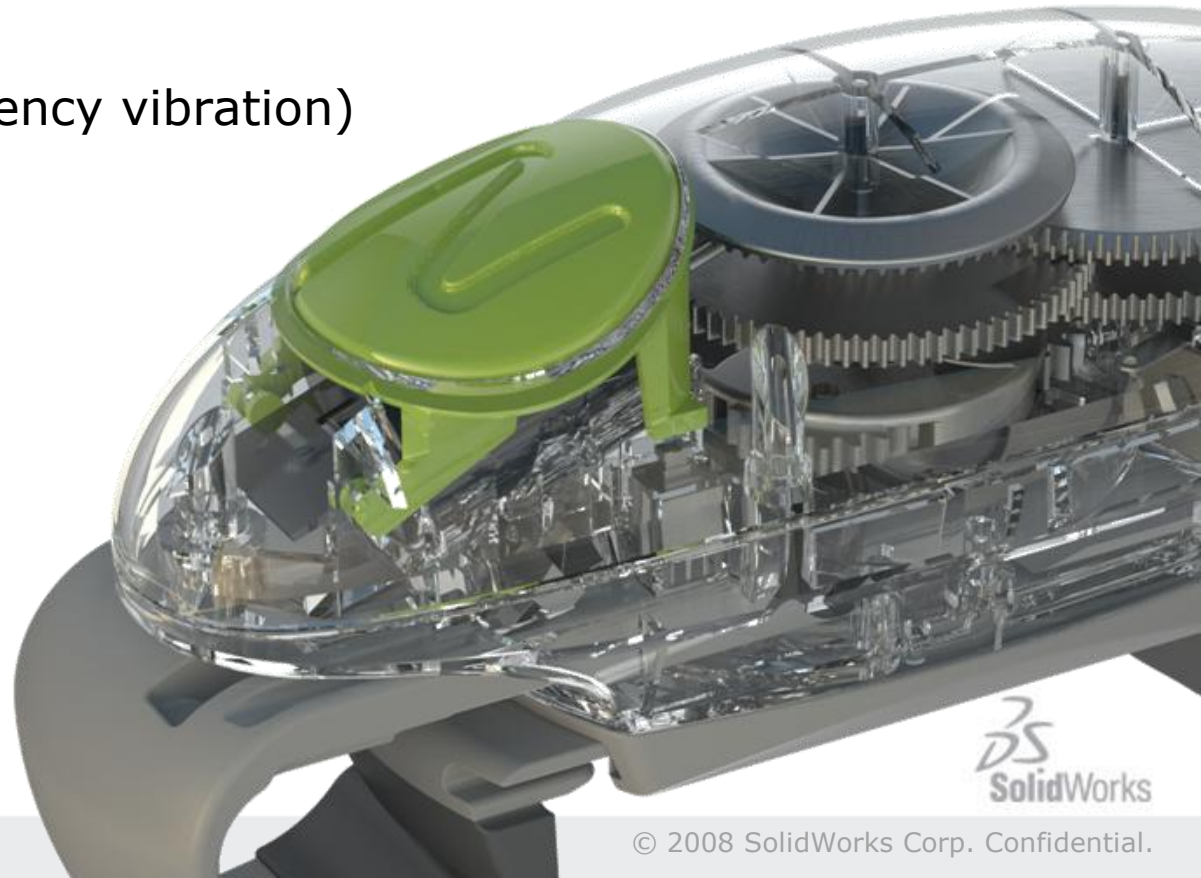
Going further – SolidWorks Premium

- Included with SolidWorks Premium
 - Assemblies – contacts
 - Many more loads and restraints
 - Customisable plots and direct output into Excel
 - Graphs for result trends
 - Connectors – bolts, springs, spot welds, bearings, pins, etc
 - Scenarios using parameters for multiple load cases
 - Shells, Beams, Mixed Mesh
 - Adaptive Meshing
 - Report writer
 - etc



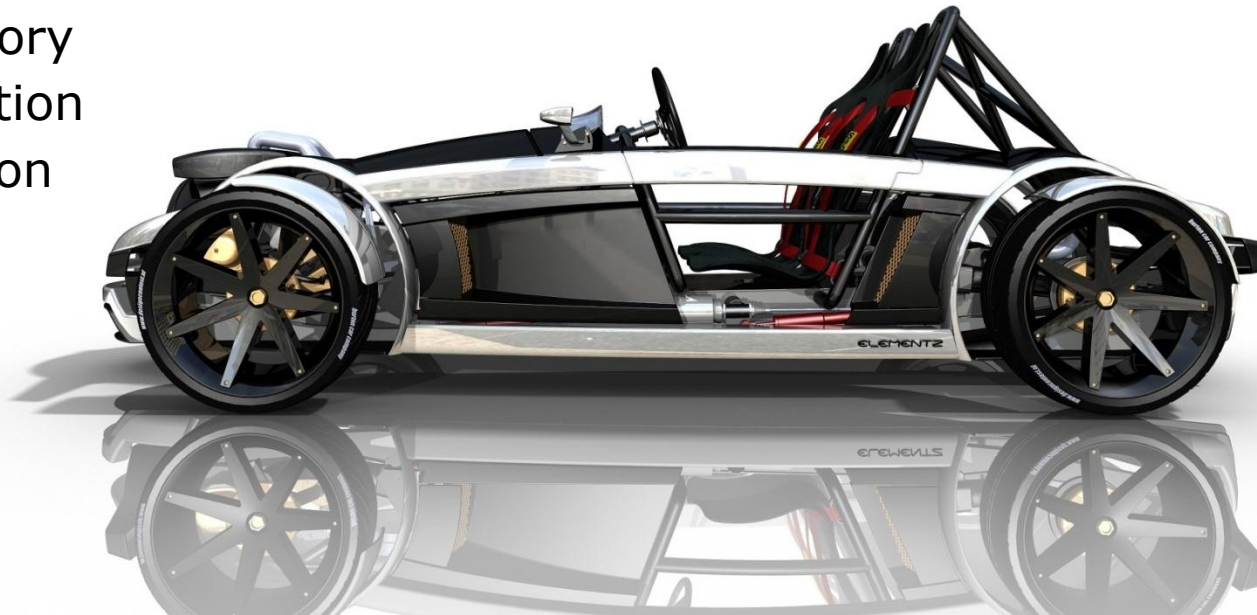
Going further - Simulation Professional

- Separate purchase
- As SolidWorks Premium plus...
 - Fatigue
 - Vibration & Buckling
 - Modal (natural frequency vibration)
 - Thermal
 - Thermal stress
 - Drop test
 - Full Optimisation



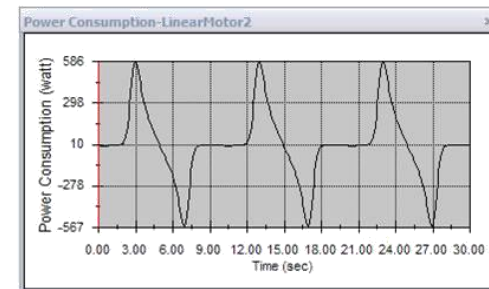
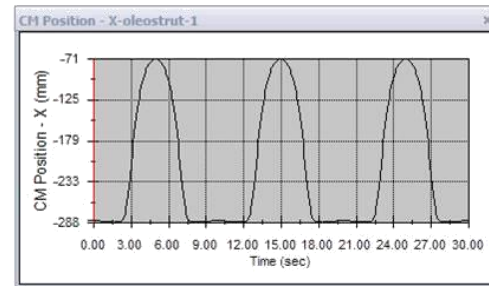
Going further – Simulation Premium

- Separate purchase
- As SolidWorks Premium and Simulation Professional plus..
 - Non-linear analysis
 - Non-linear materials
 - Non-linear loading
 - Advanced Dynamics
 - Modal time history
 - Harmonic vibration
 - Random vibration



Going further – Mechanism Simulation

- Included with SolidWorks Premium
- Kinematic and Dynamic analysis
 - Inertial loading
 - 3D contacts
 - Springs and dampers
 - Motors
 - Calculates
 - Forces
 - Velocities
 - Accelerations
 - Torque
 - Power
 - Etc
 - Integrates with other Simulation Products



Going further – Flow Simulation

- Separate purchase
- Easy to use fluid dynamics

