

**Applications of Finite Element Analysis
Ansys Workbench**

<u>Section</u>	<u>Content</u>	<u>Page</u>
1.	Course Content	1
2.	Finite Element Analysis	2-3
3.	Guidelines for successful FEA	3-6
4.	FEA Software Tools	7
5.	Usual Sequence for using Ansys Workbench	8
6.	Capabilities of Ansys Workbench Modules	9
7.	Ansys Workbench Screen Layouts and Tools	10
8.	Licencing of Ansys	11
9.	Starting to use Ansys Workbench	11
10.	Static Structural Analysis Sequence	12-15
11.	Have the Analysis Results - What Next?	16
12.	Report Generation	16
13.	Other Types of Structural Models in Ansys Workbench	17

1. Course Content & Aims

- 1.1 The course has the following aims:-
- # To appreciate the capabilities and understand the limitations of Finite Element Stress Analysis.
 - # To learn to “drive” a respected and modern FEA stress analysis system by using it to optimise the design of a mechanical component.
 - # To learn and adopt a professional approach to the use of FEA to solve engineering problems, including validation, checking and reconciliation of results.
 - # To apply all the above to a Design - Make - Test project.
- 1.2 The course includes:-
- Lectures with demonstrations of Autodesk Inventor
 - Lectures on the application of Finite Element Analysis
 - Lectures with demonstrations of Ansys Workbench
 - Supervised times of self tuition
 - Other times of self tuition
 - Public testing of components
 - Reconciliation of test results with predictions
 - Keeping of project Log Book
- 1.3 The course is intended to give a basic working knowledge of using Ansys Workbench for static structural stress analysis of 3D solid models. It does not attempt to deal with other types of models or other capabilities of “Multiphysics” in Ansys Workbench. It is assumed that Autodesk Inventor is used to solid model the components and assemblies to be analysed.
- 1.4 It is intended that each student will work through these notes, using the help facilities as required, in the process of completing the work. Attention is particularly drawn to Sections 9 - 11 of these notes that set out a typical sequence for using Ansys Workbench.

2. Finite Element Analysis

2.1 The purpose of Finite Element Analysis (all varieties) is to mathematically model a physical problem that cannot be solved satisfactorily by other means. Typical reasons for difficulties in finding solutions are:-

- that manual means of mathematical modelling cannot represent the problem sufficiently accurately
- that physical (real) models are deficient
- that full size prototypes are far too expensive

2.2 In all Finite Element Analysis, the mathematical modelling of the problem is done by dividing the problem into small pieces whose performance can be modelled simply; Finite (size) Elements. The relationships between each neighbouring element are controlled so that, taken as a whole, the “Mesh” of Finite Elements approximates to the original problem.

2.3 For small deflection, elastic, static structural analysis, the system is modelled as:-

$$[K] \{x\} = \{F\} \quad \text{where} \quad \begin{array}{l} [K] \quad \text{is the stiffness matrix} \\ \{x\} \quad \text{is the displacements (of the nodes)} \\ \{F\} \quad \text{is the forces (at the nodes)} \end{array}$$

Solving this set of simultaneous equations yields the basis of the desired solution.

2.4 Many FEA systems are now often capable of modelling time varying quantities with non-linear properties and large changes in geometry. However the core of FEA analysis, and the most reliable, is still the static structural analysis limited to:-

- elastic, homogeneous, isotropic materials
- linear material properties
- small deflections: geometry changes can be ignored
- all material well below yield: no plastic deformation

2.5 There are a number of difficulties with this approach that:-

- The elements do not perfectly represent the shape of the original problem.
- Neighbouring elements only join at their common nodes: other parts may be discontinuous.
- The actual quantities within the element are not modelled: linear or quadratic functions are used to estimate the real variations within each element.
- Such is the huge number of simultaneous equations to solve that, usually, an approximation is taken as satisfactory. The solution process iterations are usually stopped when the change from the last iteration was within the agreed amount, often 20% although 5% is also common.

2.6 The solution delivered is thus:-

- capable of completely missing important geometry, especially crucial detail,
- only intended to be good at the nodes,
- not accurate even at the nodes,
- approximate within each element with discontinuities at element boundaries,
- usually “smoothed” to make the solution appear better than it is.

- 2.7 Despite these serious “health warnings”, FEA is still extremely useful and has become an essential tool in most areas of engineering. For the integral quantities such as displacement and temperature, the errors in FEA largely cancel generally giving very accurate solutions. Local (derivative) quantities such as stress and heat flow are generally much less accurate.
- 2.8 For the reasons above, the design of the mesh (the system of elements making up the model) is very important. Even more important is the choice of modelling method and element type.
- 2.9 The whole point, usually, of determining the stresses in a component, is to determine whether the component is going to fail. Yield is only one failure mode, but a very important one. Do not forget, crack growth, buckling, fatigue, corrosion, seizure and many more.

For further information, please refer to “An Introduction to The Finite Element Method” by J C Appleby, School of Mechanical & Systems Engineering, Newcastle University.

3. **Guidelines for Successful FEA**

The methods in this section are essential to validate and check the output of FEA systems. NEVER accept FEA (or any other computer calculations) without checking very carefully!

3.1 **Validate the FEA System**

Before using on the problem intended, it is essential to try out the FEA system on a similar problem for which the answers are either already known or can be calculated by hand.

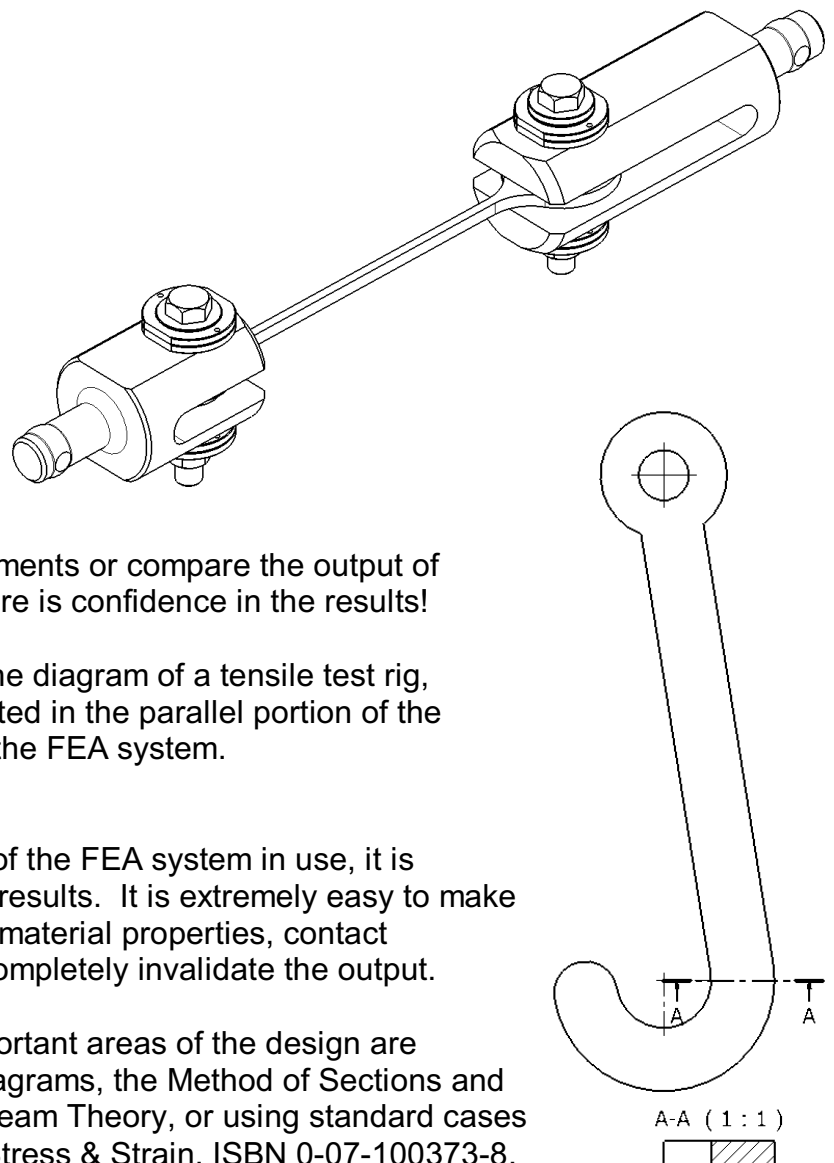
The more complicated the problem to be investigated, the more detailed the checks will need to be. In complex non-linear, time varying problems (e.g. Computational Fluid Dynamics) it may even be necessary to run physical experiments or compare the output of different FEA systems before there is confidence in the results!

As an example of validation, in the diagram of a tensile test rig, manually check the stress expected in the parallel portion of the specimen with that predicted by the FEA system.

3.2 **Manually Check the Output**

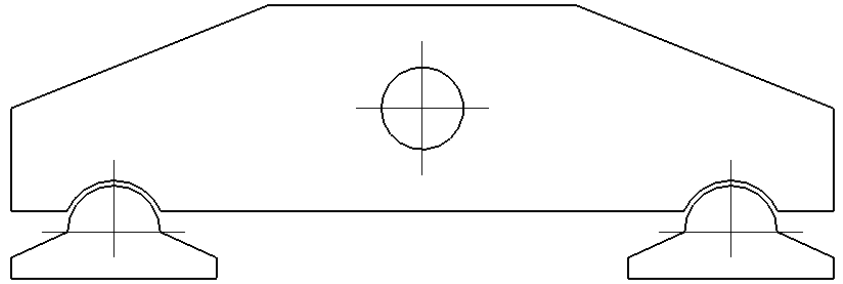
In addition to general validation of the FEA system in use, it is essential to check the individual results. It is extremely easy to make a mistake in the FEA input data, material properties, contact conditions or supports that will completely invalidate the output.

It is suggested that the most important areas of the design are checked by use of Free Body Diagrams, the Method of Sections and simple stress analysis such as Beam Theory, or using standard cases set out in Roark’s Formulas for Stress & Strain, ISBN 0-07-100373-8.



3.3 Don't Analyse the Wrong Problem

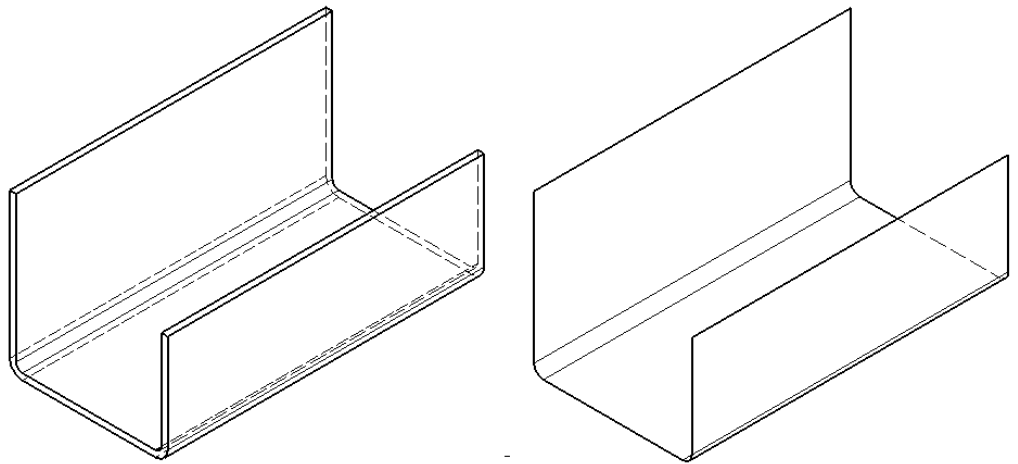
It is essential to correctly model all the constraints and contact conditions to ensure that unwanted stresses are not caused. See diagrams and THINK what deflects when parts are loaded.



In the diagram of the beam with two feet, a vertical load applied to the beam centre is reacted at the feet. The feet are intended to rotate in their sockets to allow for uneven mounting AND deflection of the beam under load. If the contact regions are specified without allowing tangential movement, a very high load will be on the inner feet corners and the contact regions will be subject to shear loads.

3.4 Don't Analyse with the Wrong Tools

As an example of using the wrong tools, consider a sheet metal bracket.



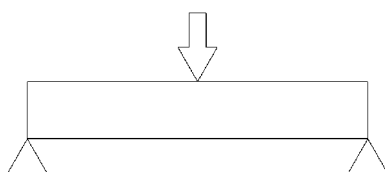
The solid model on the left will require meshing with a great number of 3D

tetrahedral elements but the surface model (IGES) on the right needs only a few plate or shell elements. Whilst the solid model might appear more accurate, the plate / shell model is simpler and will require many fewer approximations.

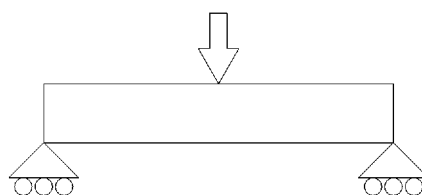
3.5 Use the Tools as Intended

3D Tetrahedral elements with mid-side nodes (10 node elements) or 20 node hexahedral elements are most common in FEA. However, they are most accurate (i.e. give the best predictions) if the elements have all three dimensions approximately equal. Variations of 2:1 usually cause few problems but above 5:1 ratios are not recommended. Similarly included angles less than 30 degrees are not recommended.

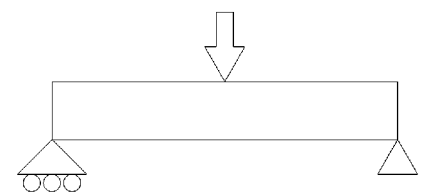
3.6 Keep your models under control



Over-constrained
Gives unwanted tensions in bottom of beam because supports cannot move.

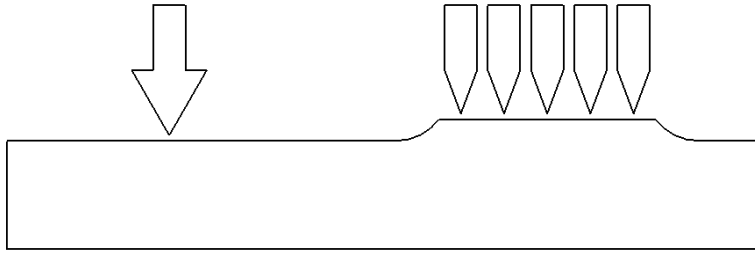


Under-constrained
Whole model may have unwanted motion. Ansys adds "Weak Springs" to deal with this rigid body motion.



Correctly Constrained
No unwanted stresses. No unwanted motion.

3.7 Apply Loads & Supports Carefully



Point Load (or Support)

Gives very high stresses:
need to ignore or
accept local plastic
deformation.

Poor model of reality

Load Applied over an Area

Need to provide defined area:
typically a pad raised 0.1mm.
Need to provide large radii to
prevent stress concentrations
and lots of unwanted elements.
Good model of reality.

3.8 Coordinate Systems & Direction of Forces / Support

Ensure that coordinate system used for loads is defined appropriately: this will usually mean that the force direction is defined by a feature on the part or assembly.

Make sure you provide a suitable feature such as a flat or cylinder to align the Force or Support vectors.

3.9 More Complexity = More Problems.

All the situations below can be modelled and solved with Ansys but require use of the special facilities: these situations seldom cause real problems, meaning that the vast majority of situations are best modelled (at least to start with) as linear, small deflection, elastic, homogeneous, etc.

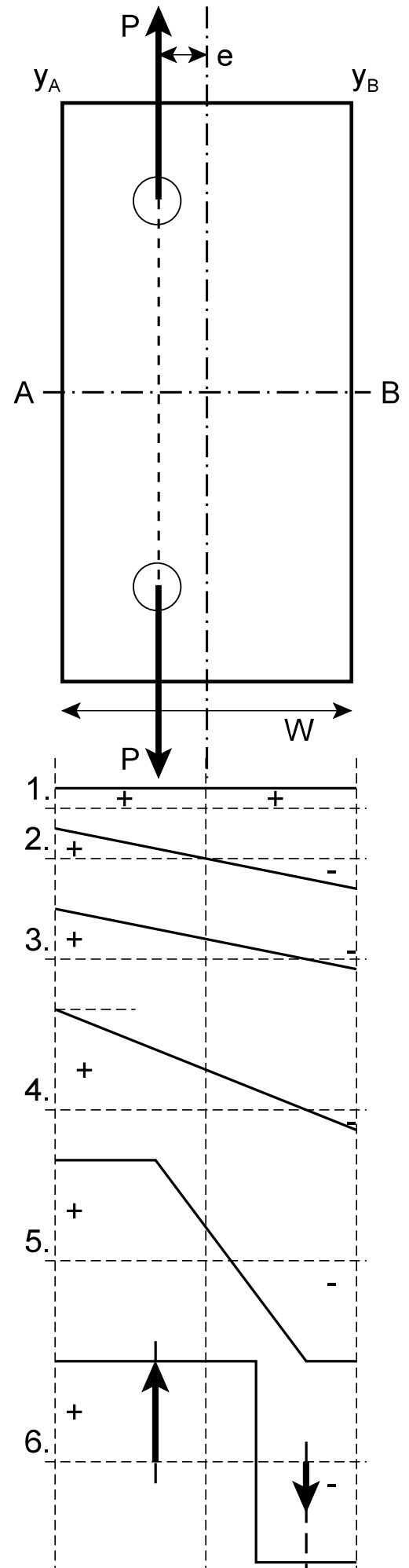
- # Contact Geometry changes with load BUT this is not normally modelled.
- # Contact Region friction alters the stresses but is hard to predict. May well be better to investigate the effects of allowing free movement ($\mu=0$) and the effects of the “bonded” case ($\mu=$ infinity), then to understand that the real situation will be between these two limiting values and as will the stresses. Use “bonded” wherever possible.
- # Plastic deformation requires use of special facilities in Ansys. However, very few components are serviceable after they have yielded so plastic analysis is not often necessary.
- # Big deflections and/or very soft materials give elastic deformations that appreciably alter the geometry of the parts: e.g. rubber bellows.
- # Non-homogeneous materials or reinforced materials can be modelled in Ansys but the material models will require especially careful validation.

3.10 Beware (common) Nasty Surprises

Most problems are more complex than they appear. Most structures, for example, have material defects, sharp corners / cracks, huge “residual stresses” from welding or production processes and are subject to fatigue and corrosion. Residual stresses can even exceed the material yield stress! The allowable stress is often a fraction of the yield stress.

3.11 Plastic Deformation

- (i) Small regions of plastic deformation are usual in engineering components. Any contacting surfaces produce deformation of the asperities (peaks).
- (ii) FE Stress Analysis is commonly limited to elastic deformation. Therefore, the mathematical model used is not correct as the plastic deformation is not modelled.
- (iii) In practice, most regions of plastic deformation are small and can be neglected without any noticeable problems.
- (iv) However, if plastic deformation (yielding) causes noticeable changes to the stress distribution, elastic (only) FEA may be very inaccurate.
- (v) Even plastic deformations are usually small in comparison to the part dimensions for the majority of components and materials: this means that the shape of the deformed component does not alter the stress distribution (significantly).
- (iv) For very flexible parts, diaphragms, bellows, springs, etc., the geometry change does affect the stress distribution greatly.
- (vii) Consider the example in the diagrams. A rectangular bar, T thick and W wide, is subjected to tension:-
 - (1 - 3) Assuming (initially) only elastic deformation, the stress on section A-B will be the sum (3) of the stresses due to axial load (1) ($\sigma_{axial} = P / W T$) and bending due to the eccentric load (2) (bending moment = $e P$)
 - (4) As P increases the stress at A just reaches yield, but B is not at yield stress yet.
 - (5) As P increases still further, side A continues to yield but the maximum stress observed is only yield stress or slightly greater, dependent on work hardening of the material.
 - (6) For very ductile materials, (e.g, mild steels, copper, polymers), the eventual stress distribution can be approximated to the final diagram.
- (viii) The load carried in (6) is much greater than in (4) but the maximum stress is unchanged (except for work hardening).
- (ix) Using the simple model of (6) it is possible to estimate the ultimate plastic deformation load. Considering a Free-Body-Diagram of either the top or bottom half of the bar:-
 - the resultant forces of each yielded area need to sum to P .
 - the moments caused by the P and the two forces need to sum to zero.
- (x) N.B.residual stresses are not include in above.



4. FEA Software Tools

There is a wide range of FEA software on the market but the most respected systems are Ansys and Nastran: each are “top” in different areas of industry. Many other companies offer easy to use FEA software that covers some market areas but few have the breath of the big two.

Ansys provides many different varieties of FEA software, including:-

- # Multiphysics FEA software for universities and research establishments that allows several different physical phenomena to be incorporated into a model at once (the version used widely at Newcastle University)
- # Design Space: a basic, easy to use, static, structural, stand-alone FEA system
- # LS-Dyna: almost the “standard” for automotive crash simulations
- # Fluent & CFX: computational fluid dynamics software

Commercially, the software is very expensive, meaning that most companies with only a passing FEA interest opt for less comprehensive packages than those available at Newcastle University.

For many of the Ansys FEA software products there are two types of user interface:-
- the traditional “classic” screen where the majority of user inputs are typed commands
- the Workbench interface where the interactions is largely icon driven
However, whether running with “classic” or Workbench interfaces, solving of the problem is performed by the same software, only the interface is different. The features of “classic” are available through the Workbench interface by inserting typed commands.

Newcastle University is running **Ansys Multiphysics** with both teaching & research licences with both the “classic” and Workbench interfaces enabled including CAD translation for Autodesk Inventor.

Each version of Ansys licence imposes limits on the complexity of the FEA models, allowing only a certain number of nodes. The *Academic Research* versions the Ansys software allow models with unlimited numbers of nodes. Both the *Academic Teaching* and the *Academic Research* versions have a “stamp” “not for commercial use”.

Referencing or Citing of work done with Ansys

It is important that the particular version(s) of software used is stated in all work, for example, "ANSYS (version) Multiphysics Academic Teaching Advanced was used..."

5. Usual Sequence for using Ansys Workbench

Before attempting FEA, it is normal to remove any features that are not of interest to the analysis or will cause problems to FEA. This is likely to include:-

- parts whose FEA is not needed because they are not in doubt,
- part features irrelevant in FEA models, e.g. threads,
- welds: modelling of welds and their effects is complex
- symmetrical or repeated features: analyse only fraction or simplest area

Typical workflows for Ansys Workbench is set out below, but other routes may be possible.

5.1	Inventor model open. Ansys WB started from Inventor menu bar & linked to CAD geometry. (3D solids & assblys)	OR	Ansys WB Design Modeler used to import CAD geometry from Inventor file. (3D solids & assblys) (3D IGES surfaces)	OR	Ansys WB Design Modeler used to create CAD geometry. (3D solids) (3D surfaces) (3D frames)
-----	--	----	--	----	--

5.2 Ansys Design Modeller (DM) generates & checks model is in suitable condition.)

5.3 Ansys Simulation takes the Inventor or Design Modeller model.

5.4 Irrelevant parts of assembly models can be suppressed.

5.5 Material of each (relevant) part defined and Engineering Data input for each material.

5.6 Contacts between (relevant) parts defined & type of contact selected.

5.7 Mesh of Finite Elements defined.

5.8 Coordinate systems (defined if necessary).

5.9 Supports defined.

5.10 Loads defined.

5.11 Types of Solution defined (e.g. Static Structural).

5.12 Model is Solved.

5.13 Results are reviewed.

5.14 If desired, changes made to the CAD model in Inventor and saved.

5.15 Model (Import) “refreshed” in Design Modeller.

5.16 In Project Tab, Simulation updated from CAD model.

5.17 Re-Solve the model and review results: repeat until satisfied.

5.18 Use “camera” to take images of results. Preview Report automatically produces report.

5.19 Generate report and ***check*** report.

6. Capabilities of Ansys Workbench Modules

ANSYS Multiphysics Academic Workbench has many programme modules and provides for many different types of analysis. Of particular note for this course are:-

6.1 Design Modeller (DM Geometry)

DesignModeler is a geometry editor of existing CAD models and a parametric feature-based solid modeler, intended for modelling 3D parts or uploading 3D CAD models for FEA preprocessing.

6.2 Static Structural (ANSYS)

This is the basic and most important FEA software tool and includes facilities to control part materials, contact regions, FE mesh, supports (constraints), loads, results and report production. This is part of the ANSYS Mechanical software.

6.3 Engineering Data

The Engineering Data modules have a library of materials data. The Engineering Data of new materials for the Project can be input or edited, but the Data in the library cannot be edited as it is part of the software installation.

(6.4 Design Exploration)

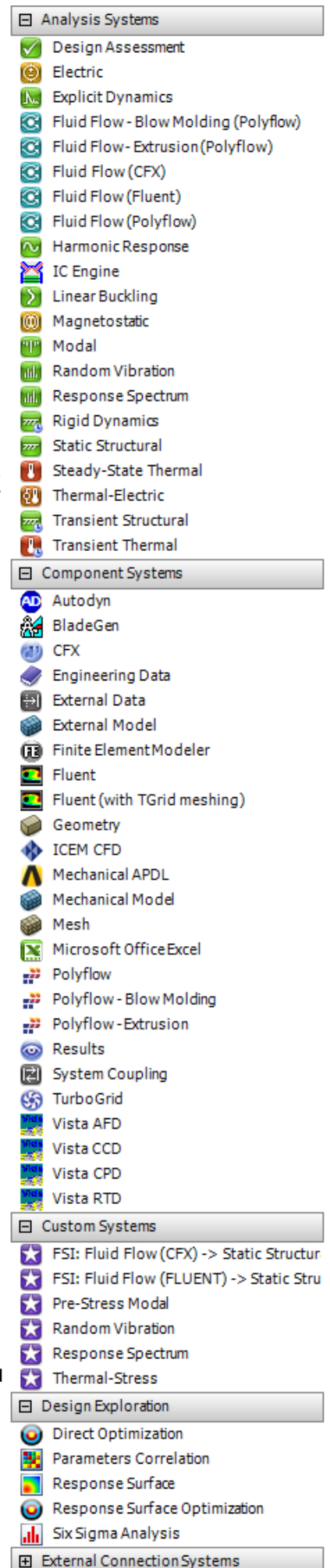
These tools enable a systematic and automatic variation of parameters to optimise designs. Powerful but not recommended for this course.

Design Explorer optimisations need to be treated with care as, although they can be amazingly effective for problems controlled by only “well behaved” functions with no step changes, step changes are common in engineering and the wrong optimum is easily presented!

6.5 Note

Ansys Workbench works with two main screens, one for controlling the project data (Project Screen) and one for the analysis (Mechanical Screen).

Use of the software is not all obvious: you are recommended to follow the sequence on the following pages closely until you are an expert!



7. Ansys Workbench Tools

Whilst there are many tools in Ansys, there are a number of tools common to more than one module of the software that assist with the viewing of the model and the selection of planes. The tools available change with the operations being undertaken.


The image displays the Ansys Workbench software interface, highlighting several toolbars and panels used for model viewing and selection:

- File Edit View Units Tools Help:** The main menu bar at the top.
- VIEW MECHANICAL OBJECT WIZARD GENERATOR:** A toolbar containing icons for Solve, Show Errors, and various view options like Show/Hide Worksheet, New, and Selection Info.
- HIT SELECT SELN. VERTEX MODE NODE EDGE BODY ELEMENT:** A toolbar for selecting different parts of the model, including Face, Box, and Rotate.
- Show Vertices Wireframe:** A panel with options for 'Show Mesh On/Off' and 'Show Coord. Systems'.
- Geometry:** A panel with options for 'Virtual Body', 'Point Mass', 'Thermal Point Mass', and 'Thickness'.
- Joint Configure:** A panel for configuring joints, including options for 'Assemble', 'Delta', 'Set', and 'Revert'.
- Visibility:** A panel for controlling the visibility of items in a group, including 'Suppression' and 'Locks'.
- Unit Conversion:** A panel for converting units, currently set to m/s².
- Edge Coloring:** A panel for controlling the color of visible edges, including 'Thicken Annotations'.

8. Licencing of Ansys

- 8.1 As Ansys is highly valuable software, the suppliers go to great lengths to avoid unauthorised use: the versions in use by Newcastle University would cost commercially several tens of thousands of pounds per seat and also need an annual fee to use.
- 8.2 All the Ansys educational licences for Newcastle University are “served” from the central licence server computers in Claremont Tower. There are a limited number of “Academic Research” licences and a much greater number of “Academic Teaching Advanced” licences. The “Academic Teaching Advanced” licences permit element mesh / node numbers more than sufficient for all teaching related projects: use these licences.

9. Starting to use Ansys Workbench

- 9.1 Ansys workbench may be started from the usual Windows - Start - Programs menu, or if Autodesk Inventor is running, it may be invoked **Ansys** Tab of Inventor by clicking **Workbench** (yellow icon with black upwards arrow ).
- 9.2 Ansys Workbench has its own geometry modeller / CAD software called Design Modeller (**DM**). DM can generate 2D and 3D designs but is not nearly so comprehensive as Inventor and is much less easy to use. However, there are some (few) modes of using Ansys Workbench that requires the geometry to be generated in DM: e.g. if beam elements are required, DM frame geometry is needed. For the vast majority of projects, it is recommended that Inventor is used to generate the models.

Either method of starting leads to a very similar screen.

- 9.3 Not all of Ansys Workbench is intuitive! The following welcome message helps...

Welcome to ANSYS Workbench!

If you're new to ANSYS Workbench, we strongly encourage you to take just a few minutes to view some [tutorials](#).

If you prefer to get started immediately:

1. Select your desired analysis system from the Toolbox (at left), drag it into the Project Schematic (at right), and drop it inside the highlighted rectangle.

2. Right-click on the Geometry cell to create a new geometry or import existing geometry.

*3. Continue working through the system from top to bottom. Right-click and select **Edit** on a cell to start the appropriate application and define the details for that part of the analysis.*

*As you complete each task, a green check mark appears in the cell, indicating that you can proceed to the next cell. ANSYS Workbench automatically transfers your data between cells. When you select **Save** (either from the ANSYS Workbench window or in an application), the entire project is saved.*

You can connect systems to build more complex projects.

For details, see [Working in ANSYS Workbench](#).

*Press **F1** at any time for context-specific help.*

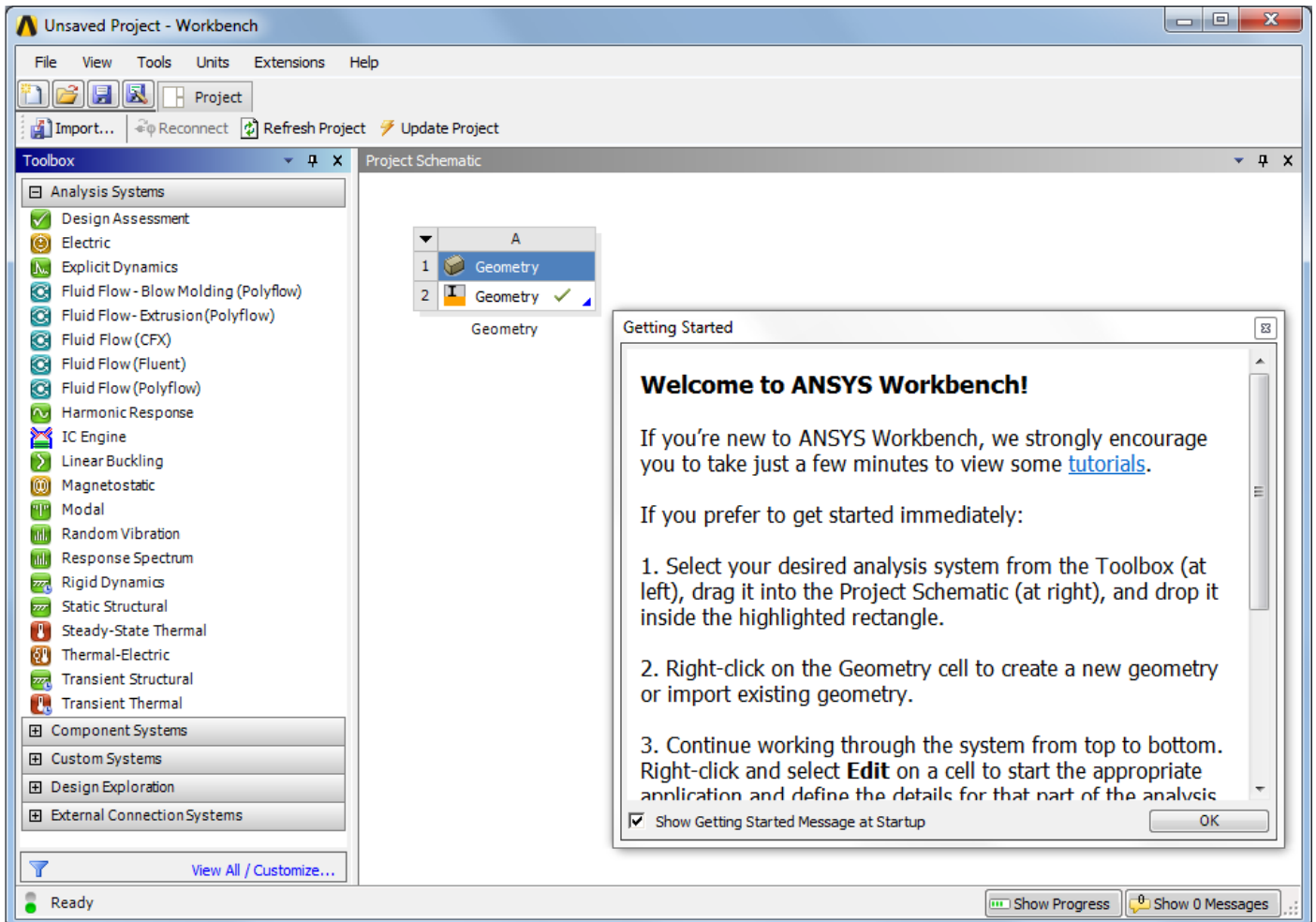
Ansys has a large collection of on-line tutorials and local help files. Some are even helpful!

10. Static Structural Analysis Sequence

10.1 Starting Ansys (from Inventor or direct)

In Inventor -> **Ansys** Tab -> **Workbench**

After seeing a graphic during start-up, the following screen appears:-



Read then close the **Getting Started** window.

Drag the **Static Structural** tool to Cell A2 **Geometry**.

The Geometry is that already attached / opened by Ansys, i.e. the currently open Inventor Part or Assembly.

If the Geometry is not yet linked to the Ansys Project, drag **Static Structural** to the dashed green rectangle below Cell A2 and in the new Table (B) that is created.

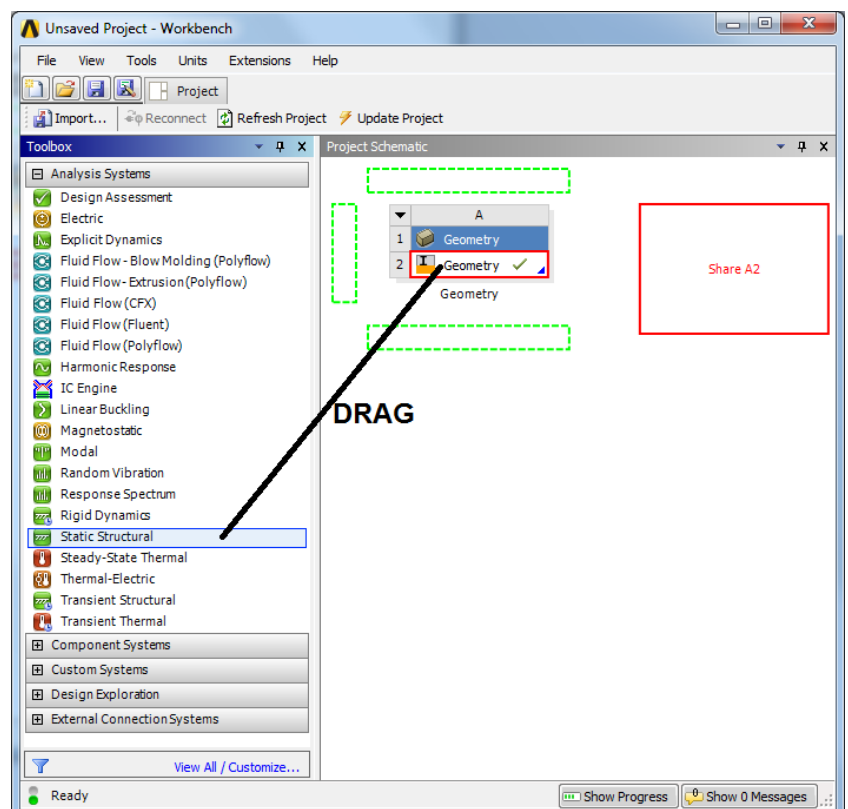
-> Right Click **Geometry**

-> **Import Geometry**

-> selected CAD file open.

CAD files are now in Ansys Project.

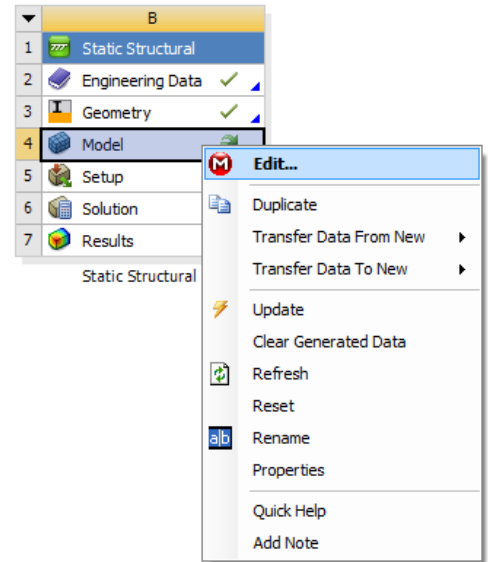
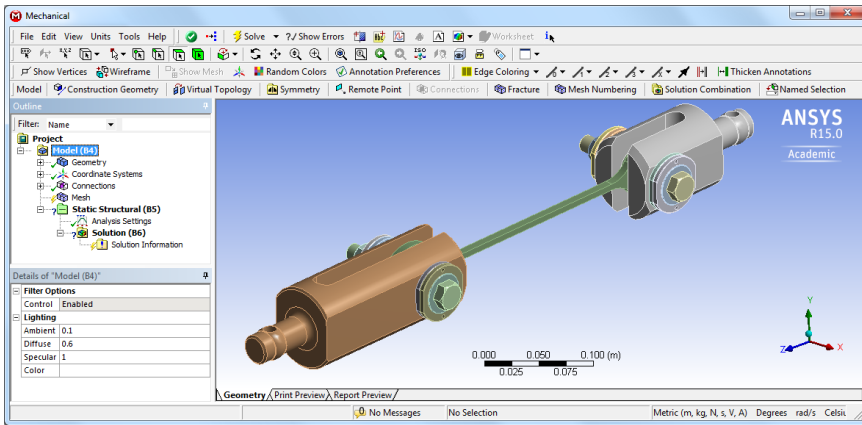
Save Ansys Project to the same folder used for the Inventor files.



Right Click **Model** -> **Edit**

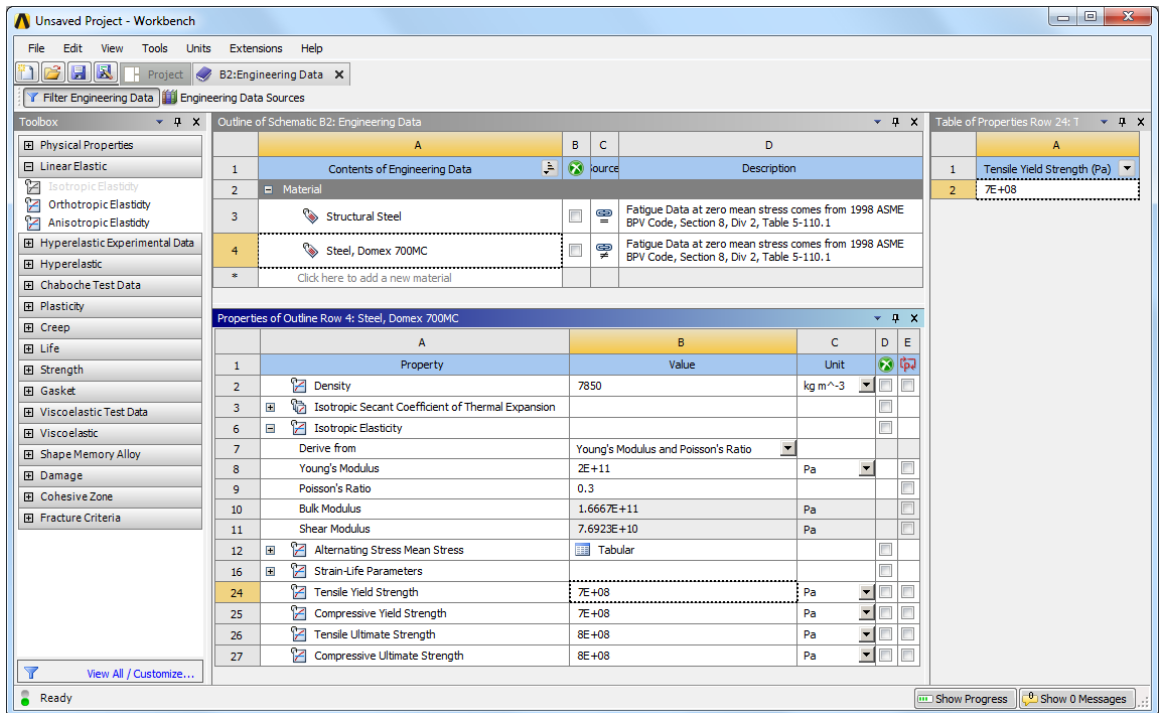
Ansys then starts the **Mechanical** window: this takes time.

Whilst the **Mechanical** window is starting, the CAD files are being “attached” to the Ansys Project.



10.2 Inserting Material Data

Initially only “Structural Steel” is available as a material for each part in the **Detail Panel** of Ansys Mechanical. Other materials are likely to be needed: probably their properties will be similar to some of the materials already in the Ansys database. The aim of the following sequence is to get an appropriate material (e.g. Domex 700MC steel) available for analysis.



In the **Project Schematic** Pane of the **Project** Window

Right Click **Engineering Data** in Box B2 (Table B, Row 2) -> **Edit**

In **Contents of Engineering Data** Pane

Right Click **Structural Steel** -> **Duplicate** (Creates “Structural Steel 2”)

In **Contents of Engineering Data** Pane

Click **Structural Steel 2** -> **Edit** it to **Steel, Domex 700MC** (or as requ.)

Click **Steel, Domex 700MC** to select it.

In Pane below with **Property** list, Edit the following values in the Domex 700MC table to:-

- Tensile Yield Strength, 700MPa (type 700e6)
- Compressive Yield Strength, 700MPa (700e6)
- Tensile Ultimate Strength, 800MPa (type 800e6)
- Compressive Ultimate Strength, 800MPa (800e6)

Click **Project** button to close Engineering Data Windows

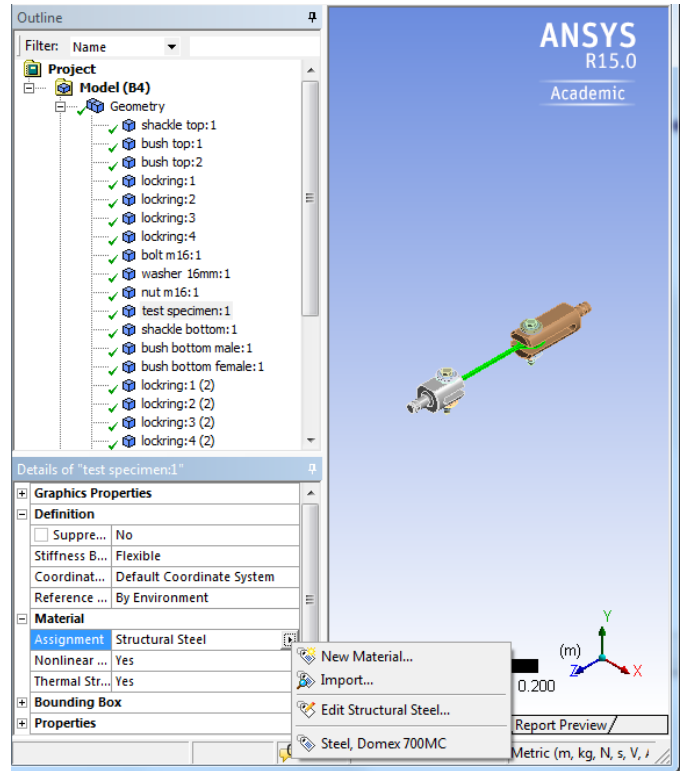
Click **Refresh Project** to make new material available.

Save the Project.

10.3 Preparing the Model Geometry

The **M Static Structural** screen (see Section 10.1) has a browser panel named Outline. In this panel is a tree of items relating to the model: preparing the analysis is essentially a matter of working down each branch of the tree until all is correctly prepared to run (“Solve”) the FEA analysis.

- (i) The first branch to prepare is the **Geometry**.
- (ii) If an assembly model, it may be that analysing all the components in the model is not necessary and will just slow the computer down.
- (iii) Expanding the geometry (click + to left of Geometry) allows all the components to be controlled.
- (iv) Right clicking the components enables each to be **Hidden** (but still calculated) or **Suppressed** (left out of the calculations). It is usually best to suppress all components not required for an accurate result.
- (v) Selecting an individual component in the Geometry tree shows its details in the **Details** box.
- (vi) **Details**, including **materials**, can be edited for each component. Just click on the detail to edit. (If appropriate materials are not visible when the material detail box is selected, follow the sequence in Section 10.2)



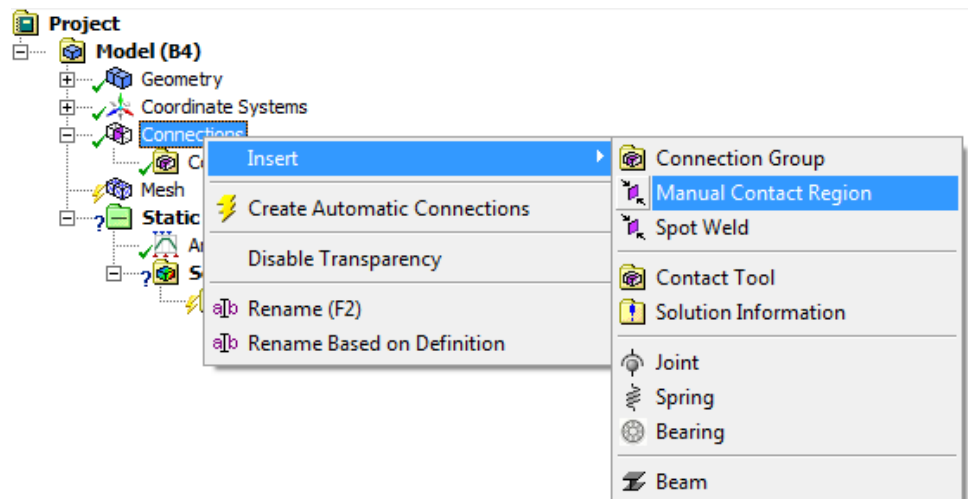
10.4 Coordinate System

If the coordinate system is not as required, expanding and right clicking the Coordinate system branch allows a different coordinate system to be defined.

10.5 Inter-Component Connections

Ansys automatically assumes there is a connection between components that are closer than the “Pinball” diameter: the default Pinball diameter works well.

The connections (joints) between components can be suppressed (i.e. neglected completely) or used for the analysis.

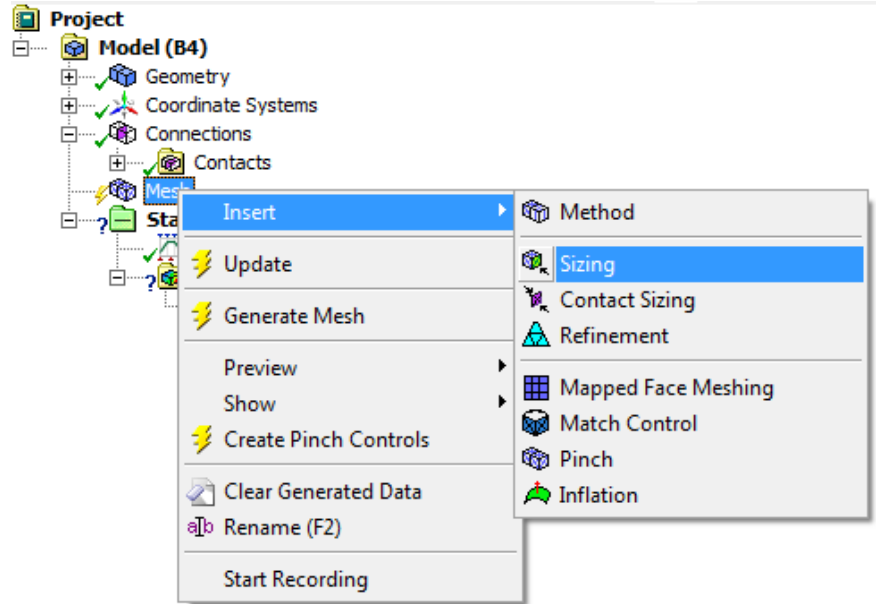


Each joint, in the Detail boxes, can be defined as **Bonded**, **No Separation**, **Frictionless**, **Rough** or **Frictional** or **Forced Frictional Sliding**. Bonded is simplest to calculate.

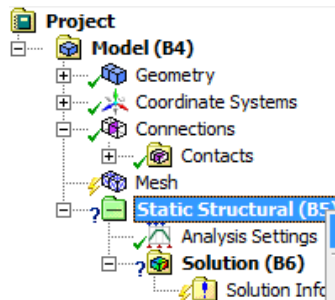
Other types of elastic or constrained connections (contacts) are also available.

10.6 Finite Element Meshing

- (i) The default finite element mesh generated by Ansys is usually sensible.
- (ii) However, it is common to need “**mesh refinements**” (to reduce the element size) or other changes to the mesh to improve accuracy.
- (iii) The mesh alterations may be inserted by right clicking the **Mesh** branch.



10.7 Loads & Constraints

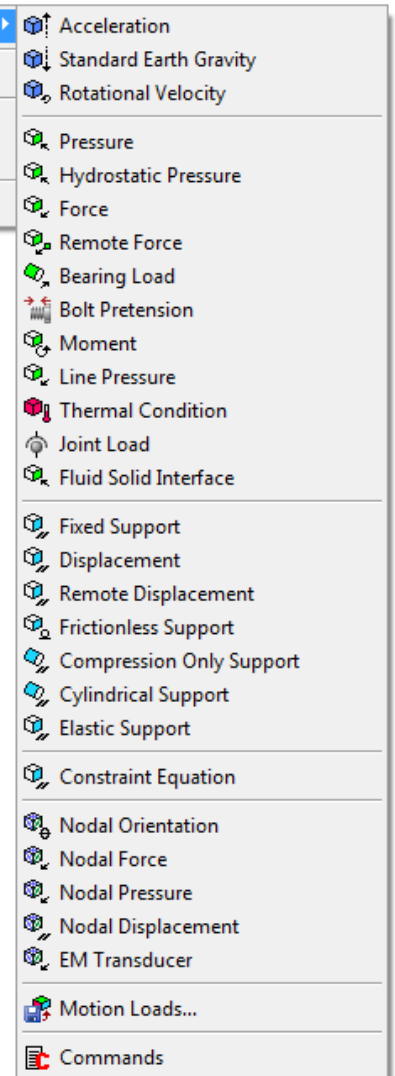
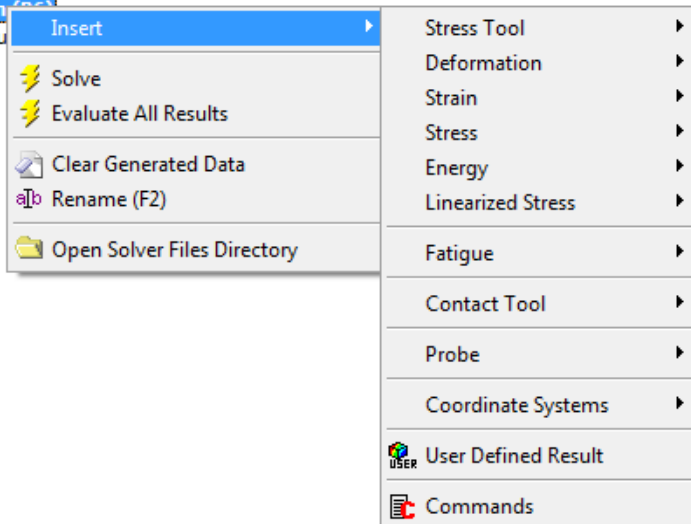
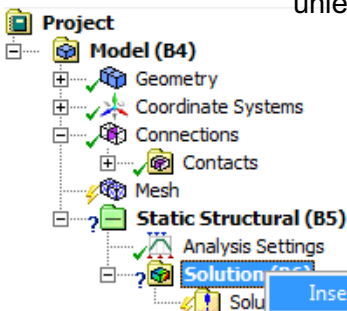


The loads and constraints can be added to the model by right clicking on the Static Structural branch.

10.8 Solution

Even if Ansys calculates (“Solves”) the model, unless there are **Solutions** specified, no results will be visible.

Right click on Solutions for results.



11. Have the Analysis Results - What Next?

- 11.1 Check the results carefully. Make sure that the analysis is correctly modelling the real situation. If results are not representative of reality, modify the model or analysis.
- 11.2 For every modification to the model / analysis, record the model, analysis & results (in your log book).
- 11.3 Even if the model is OK and sufficiently “real”, check carefully to ensure that the FE mesh does not miss any important small detail or give unwanted and spurious results. Modify the mesh as necessary and/or increase the number of iterations (convergence).
- 11.4 If the modelling AND analysis are good, the design of the components might need attention. If so, return to Inventor, modify as necessary and save the CAD model. Then in Ansys Project Schematic, right click on Geometry and select “Update”. Then re-run (Solve) the analysis.
- 11.5 If the results are just as expected and the design performing as hoped, there is no need to modify the design or the FEA model. This likely to be a very rare situation on the first run! If this is the case, proceed to the Report generation process.
- 11.6 The **Probe** tool is particularly useful for checking detail.

12. Report Generation

- 12.1 ONLY ONCE YOU HAVE CHECKED AND ARE SATISFIED that:-
 - the aspirations of the customer were correctly understood,
 - the problem was correctly modelled: loads, material, interfaces,
 - that, from your check of Ansys Workbench, it gives accurate results,
 - the results are correct,

Only then:- (see Section 7 for location of tools)
Use the **Label** and other tools to include relevant notes on each solution.
Use the **Figure / Image** tools to record images of each solution.
(these images will appear in the browser tree)

- 12.2 The **Preview Report** tab at the bottom of the Graphics Area is used to set up automatic report production.
 - Fill in the appropriate details.
 - Select appropriate figures / illustrationsAt bottom of form,
 - Click the **Report Preview** Tab at bottom of graphics area.

The report produced is a compilation of all the information Ansys has about the project. If the data needs editing, it must be edited in the database (not the report) and then the analysis re-run with the updated data.

It is likely that the report information is useful but not appropriate to be handed directly to the customer without a lot of careful formatting of the report.

Once correct...

- Save the file and report.
- It may also be **Published** to an HTML file.
- Check the report very carefully before sending it anywhere!

13. Other Types of Structural Models in Ansys Workbench

13.1 3D Solid Models (as referred to in this document)

3D solid models are appropriate for FEA work on a wide range of 3D CAD models. The choice of element shape (type) will generally depend on the purpose of the analysis and the shape of the model. Many FE models use 10 node tetrahedra or 20 node hexahedra but the number of elements becomes uneconomically large if very thin features (in comparison to the rest of the model) are used. This is because the elements, would be very distorted to fit into very thin areas if they were not to be too small. Worst of all to model are thin tapering slivers. For thin parts or features of parts a different shape is best, usually with the elements being blocks of the same thickness as the part / feature. Ansys Workbench 12.0 automatically chooses element shape to suit the geometry. However, no finite element mesh can perfectly model a design, by definition: be careful!

3D solid models can be directly imported as Inventor Part files (*.IPT) or Inventor Assembly files (*.IAM) in Ansys Workbench. They can also be used directly from Inventor, invoking Ansys Workbench from the Inventor menu bar.

However, it may not be sensible to model hollow box structures and is NOT sensible to model frameworks by this method.

13.2 3D Shell / Plate Models

Complex thin 3D forms are sometimes modelled as thin (flat) plates or (curved) shells. Ansys has elements especially for this work: these save a great deal of computation but are limited to elements of plates / shells of constant thickness and bending properties.

Ansys Workbench can import the surface models in *.IPT files. It is necessary to specify the thickness of the plate / shell in Ansys. If complex IGES files (*.IGS) are used the surface model may develop small gaps which then need "healing" before the model can be analysed.

This method may be best for complex thin box-like structures or thin curved hollow forms.

13.3 3D Frameworks

Frameworks can be modelled with Beam Elements joining nodes. The cross section of each beam element is controllable but constant between nodes. 3D sketch geometry cannot be imported from Inventor, so the geometry must either (if possible) be "picked off" 3D solid geometry that can be imported and then modified, or drawn from scratch in Design Modeller.

This method is best for space-frame type structures.