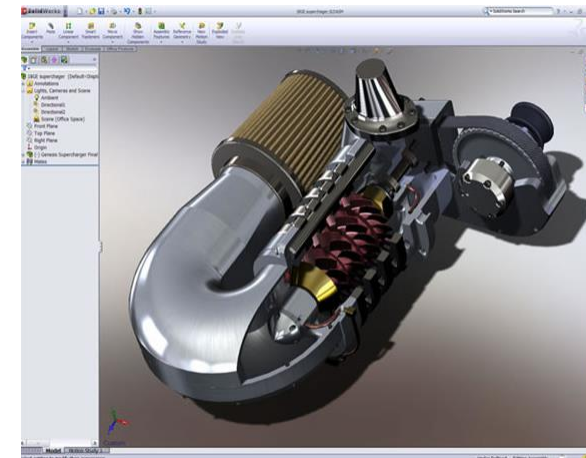
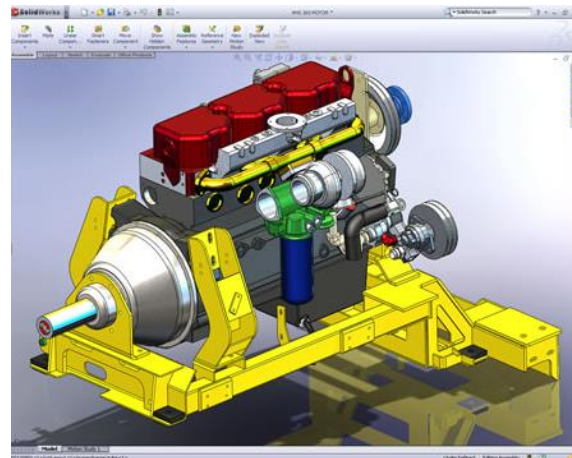
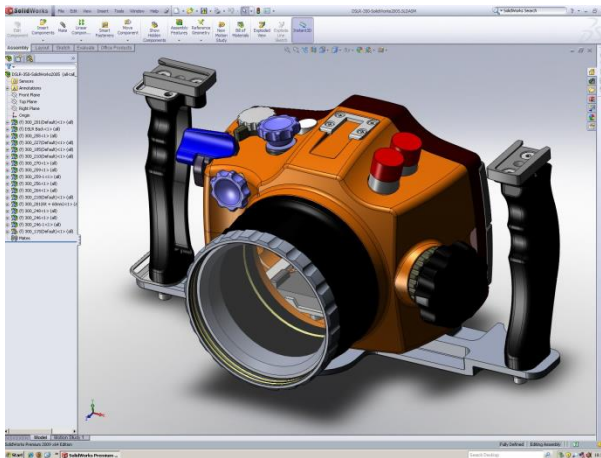


SolidWorks Training

**Introductory course for staff and students
from the School of Physics and Astronomy**



Overview

- Introductory presentation about SolidWorks
- Tutorial on creating a part
- Tutorial on building an assembly of parts
- Tutorial on exporting files
- Presentation on Engineering drawings
- Tutorial on creating engineering drawings

If you are a beginner by the end of this course you will have the foundations to go and comfortably use SolidWorks to create parts, assemblies and drawings.

If you are a false-beginner you will learn some new or alternative techniques to create parts and assemblies and ensure that they are well defined and can be easily edited if you require to change them at any time.

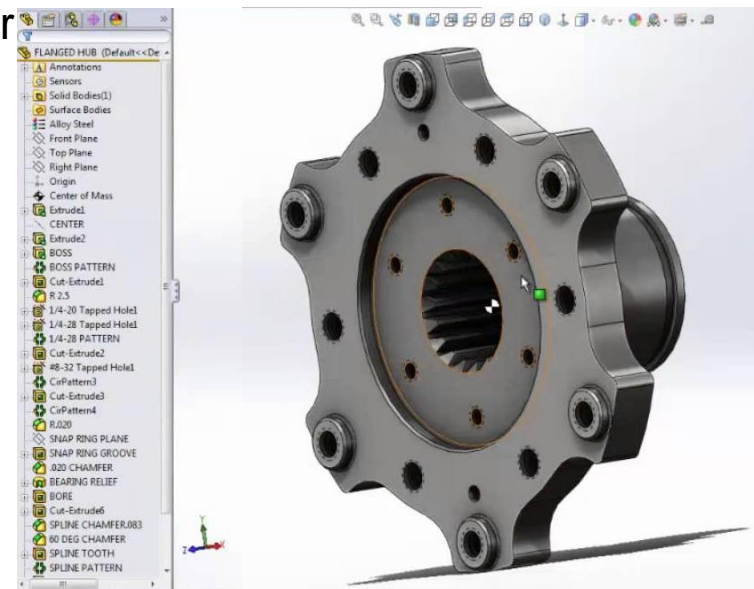
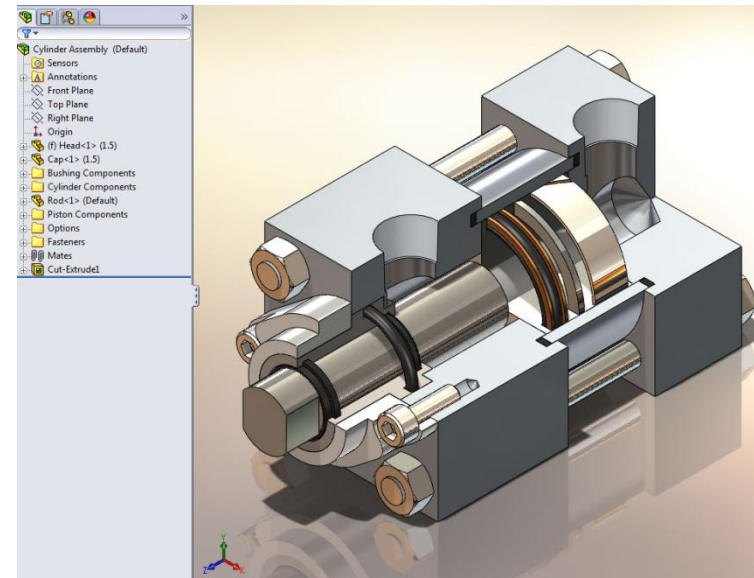


SolidWorks is a 3D solid modelling package that:

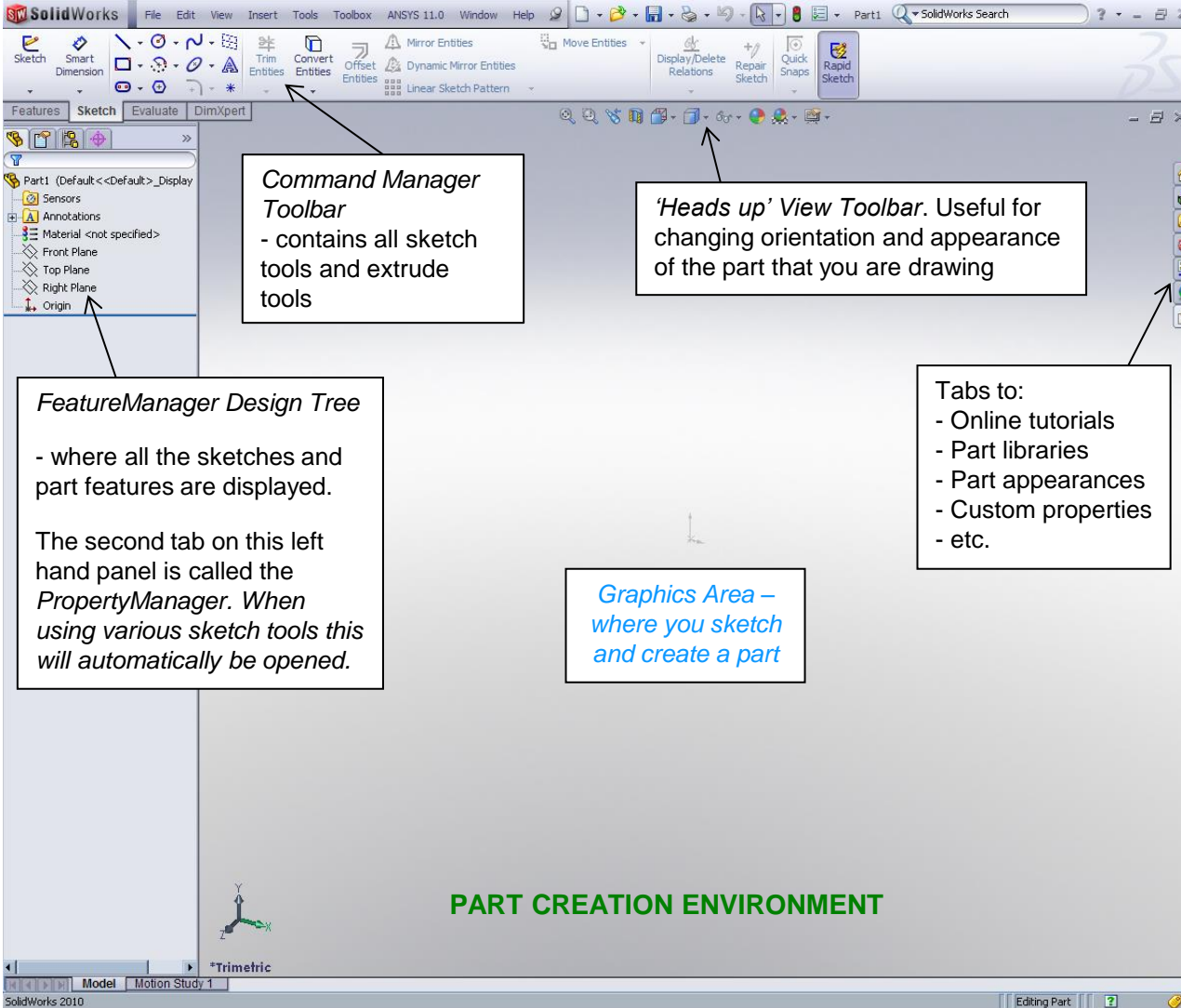
- can be used to produce three dimensional solid parts and assemblies to give accurate, scaled visual representations of parts and mechanisms.
- can be used to produce 2D manufacturing drawings
- can be used to produce solid parts that can be exported for use in ANSYS/ & other finite element programs or CAM software.
- is 'feature-based' such that parts and assemblies can be easily revised and updated as part of an iterative design approach.
- has a multitude of tools and add-ins to cater for a wide variety of industries, (many of which I wont cover today!), e.g.
 - Solidworks Simulation - Finite Element Analysis
 - SolidWorks Routing – for mapping electrical cable paths
 - Rendering (PhotoView 360)

Solidworks is not a 2D drawing package:

- Like AutoCAD, so its not great for electronic schematics or architectural layouts (but you can do them!)



SolidWorks Graphical User Interface (GUI)



The screenshot shows the SolidWorks 2010 interface. At the top is the menu bar (File, Edit, View, Insert, Tools, Toolbox, ANSYS 11.0, Window, Help) and a search bar. Below is the Command Manager toolbar with various sketching and modeling tools. The left side features the FeatureManager Design Tree and PropertyManager. The main area is the Graphics Area, and the bottom right has a vertical toolbar with tabs for online tutorials, part libraries, and custom properties. A 3D coordinate system is visible at the bottom left.

Command Manager Toolbar
- contains all sketch tools and extrude tools

'Heads up' View Toolbar. Useful for changing orientation and appearance of the part that you are drawing

FeatureManager Design Tree
- where all the sketches and part features are displayed.

The second tab on this left hand panel is called the *PropertyManager*. When using various sketch tools this will automatically be opened.

Graphics Area – where you sketch and create a part

Tabs to:
- Online tutorials
- Part libraries
- Part appearances
- Custom properties
- etc.

PART CREATION ENVIRONMENT

More detail on
Page 2 of the
Tutorial...

Using SolidWorks

The SW interface is designed for use in Windows and has similarities to the interface in many Microsoft products.

The mouse

- 1. Left button – selects items, starts sketches, closes sketches, etc
- 2. Right button – opens up a pop-up menu with additional functions for what the cursor is pointed to
- 3. Middle button
 - Rolling – Zooms in and out
 - Pressing – allows the part to be rotated

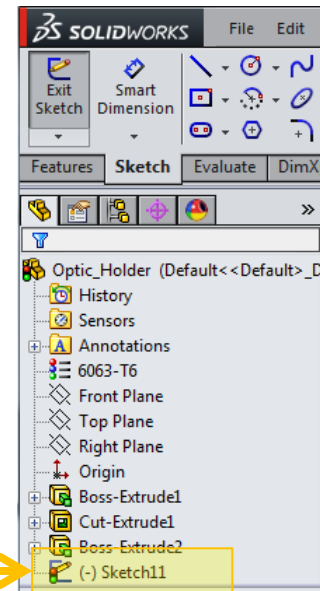


Sketch Definition

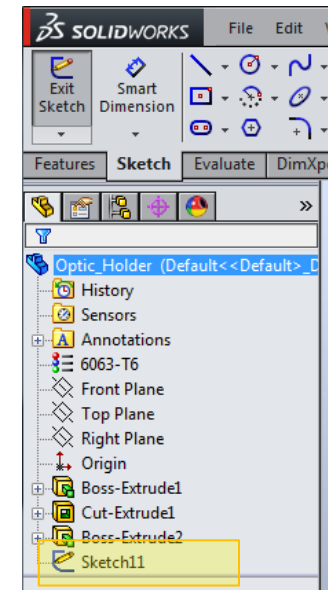
- Lines are blue until they are fully defined with relations or dimensions.
- Lines turn black when they are fully defined
- When a sketch is not defined, it shows up with a (-) sign next to it in the *FeatureManager Design Tree* (left hand panel)
- To edit a sketch that you have closed, double click on it in the *FeatureManager Design Tree*

Useful keyboard shortcuts

- Esc – exits the tool that you are using or step that you making (a quick way to get out if you have started a sketch and it is wrong!)
- Ctrl or shift – like many programmes, Ctrl or Shift can be used to selected multiple items.
- Space bar – pulls up a 'Views' menu such that you can select an orientation to view the part/assembly that you are creating.
- S – When sketching, this pops up a small menu with commonly used sketch tools at the point where your cursor is.



Sketch not fully defined



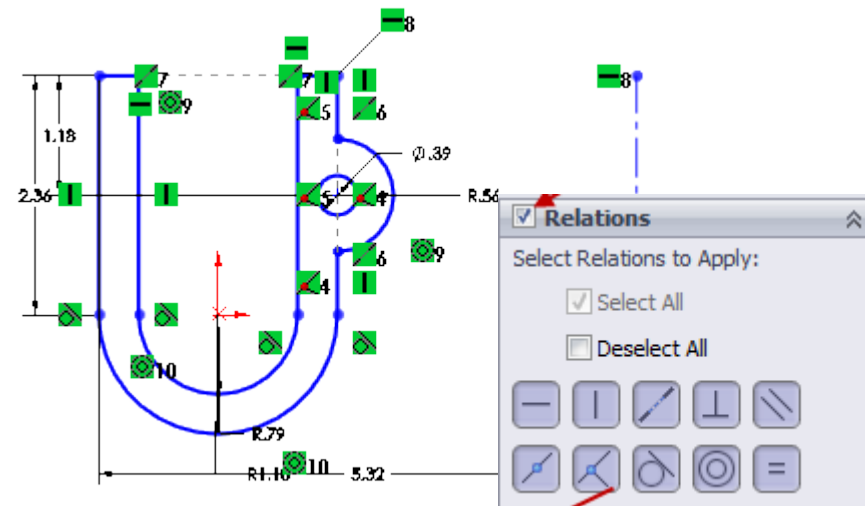
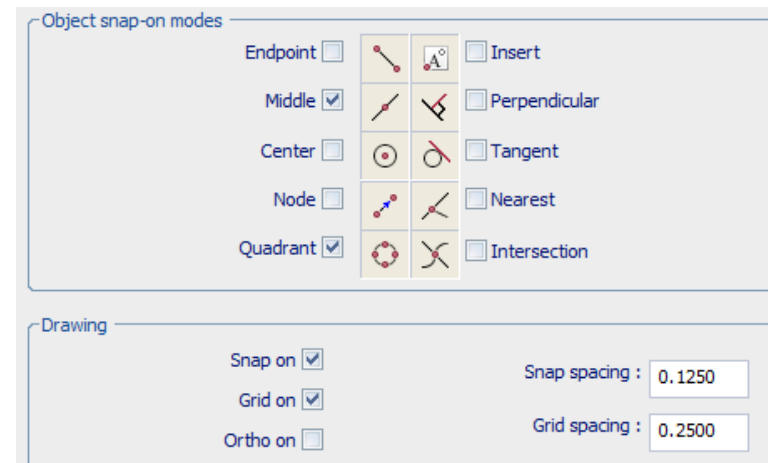
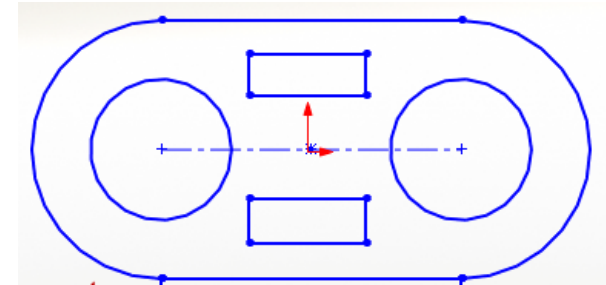
Sketch fully defined

Things to know before starting

- Use the sketch origin
 - it is often useful at a later stage for manufacturing but is also useful just to keep things tidy
 - If your part is symmetrical – sketch with the origin at the symmetry plane
- Snaps
 - In sketch mode when creating parts and when hovering the cursor over any existing sketches, or features, ‘snap’ points will be highlighted, e.g.
 - At the corners of a square
 - At mid-points of line
 - At quadrant points on a circle
- Relations
 - In sketch mode when creating parts, points can be made to ‘relate’ to each other, e.g.
 - Horizontally
 - Vertically
 - Concentrically
 - Perpendicularly

And, in assembly mode:

- Mates
 - In assembly mode, parts are ‘mated’ to each other to define their position with respect to one another, e.g.
 - Flat Surface to flat surface
 - Cylindrical surfaces concentrically aligned



Last of all, before we get started...

Customised SolidWorks Tools

- There are a few custom tools that have been designed specifically for those in the School of Physics and Astronomy that add useful detail to parts and assemblies, and allow for the smoother creation of engineering drawings.
- Before we start the tutorial we will install them on to your computer, and point SolidWorks to look at them.
- These custom tools can be from the following location:
 - <http://www.physics.gla.ac.uk/~mplloyd/Eng-SW/>
 - Download the file: *UGL_SW_Templates_Etc.zip* to a new folder in my documents called:
 - C:\Users\yourname\Documents\Temp\

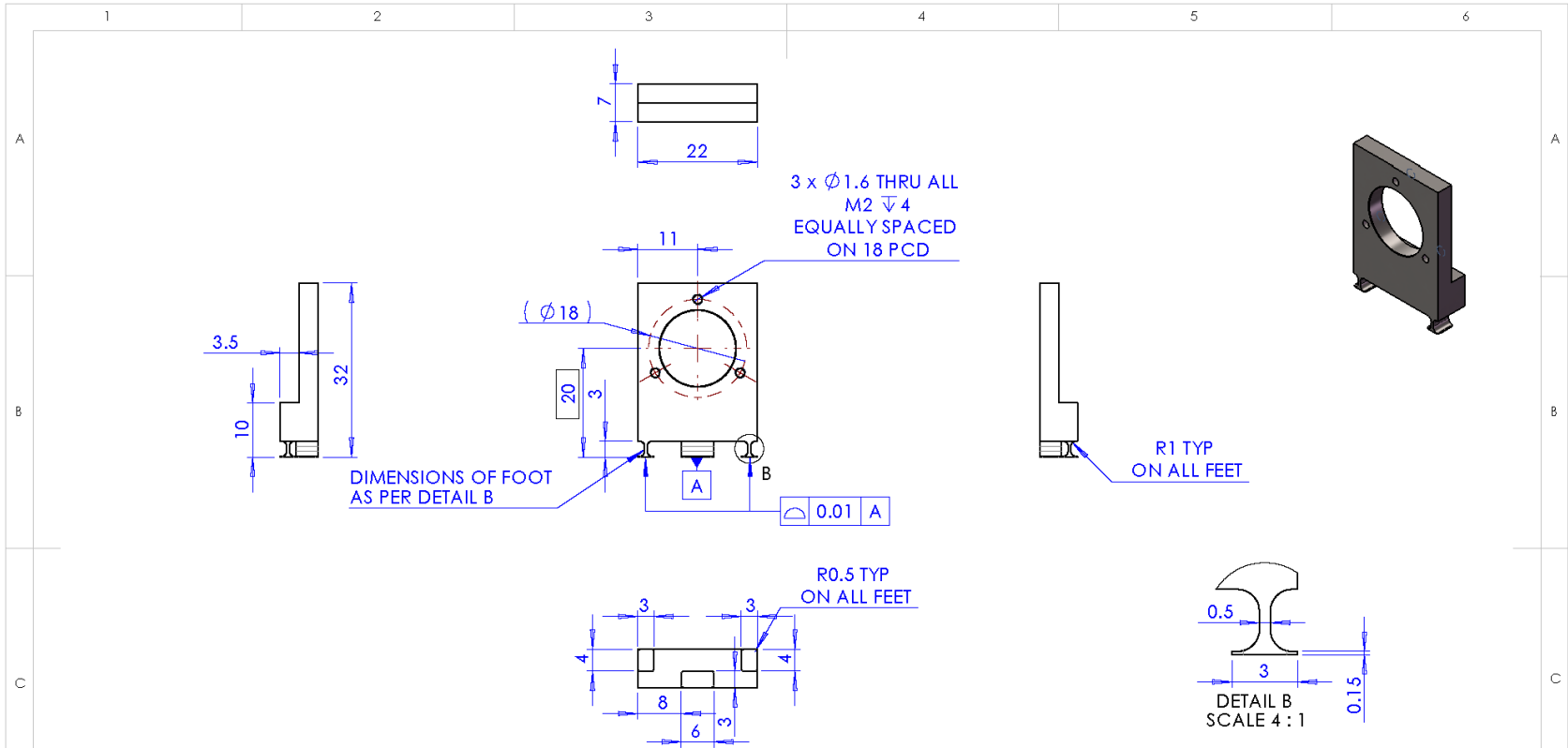


Tutorial One – Parts

Tutorial Two – Assemblies

**Tutorial Three – Exporting Parts or assemblies for use in
other CAD packages/FEA/CAM**

Engineering Drawings



NOTES (UNLESS OTHERWISE SPECIFIED):

DIMENSIONS ARE IN MILLIMETERS


GENERAL TOLERANCES:
 0<X<50mm: ± 0.1 mm
 50<X<150mm: ± 0.2 mm
 100<X<1000mm: ± 0.3 mm
 ANGULAR: $\pm 0.2^\circ$

GEOMETRIC TOLERANCES:
 ISO-2768-K

1. DEBUR AND BREAK SHARP EDGES
2. 3D CAD MODEL OF PART TO BE SUPPLIED

QUANTITY: 5

MATERIAL:
 Titanium (ASTM Grade 2)

SURFACE TEXTURE (μm): 3.2
 UNLESS STATED

FINISH:
 Clean of Oils

NAME	DATE
DRAWN M.Perreux-Lloyd	02MAR11
CHK'D	
APPV'D	

DWG NO. LOB-UGL-DRW-3077

SCALE: 1:1

DO NOT SCALE DRAWING

A4

 **University of Glasgow** Department of Physics and Astronomy
 Institute for Gravitational Research

PROJECT: LISA

SUB-SYSTEM: EBB

ASSEMBLY: Small Waveplate Holder

PART NAME: Small Waveplate Bracket

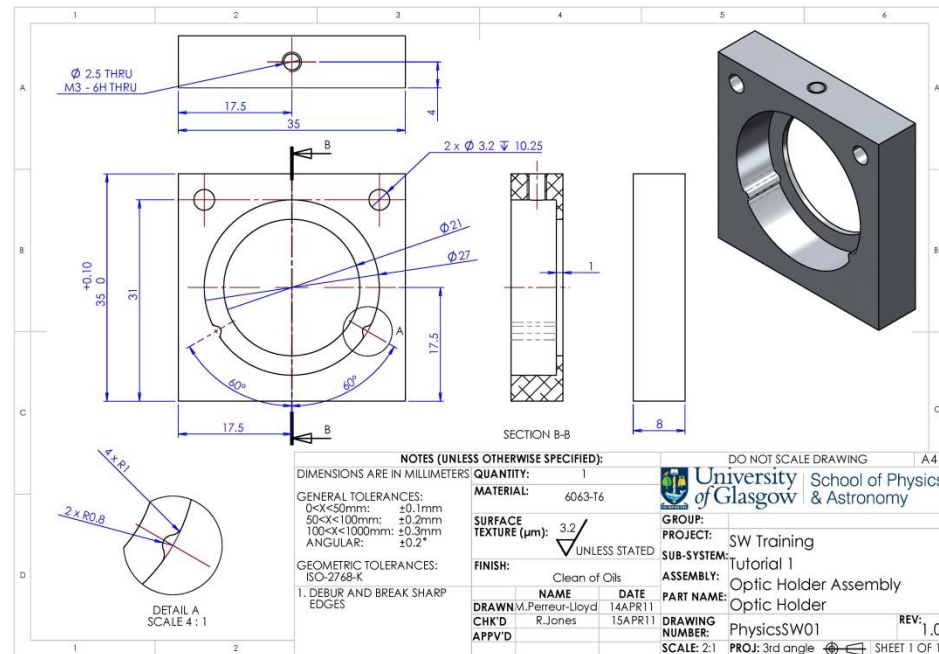
REV: 00

PROJ:   SHEET 1 OF 1

Engineering Drawings (1)

General rules when making an drawing:

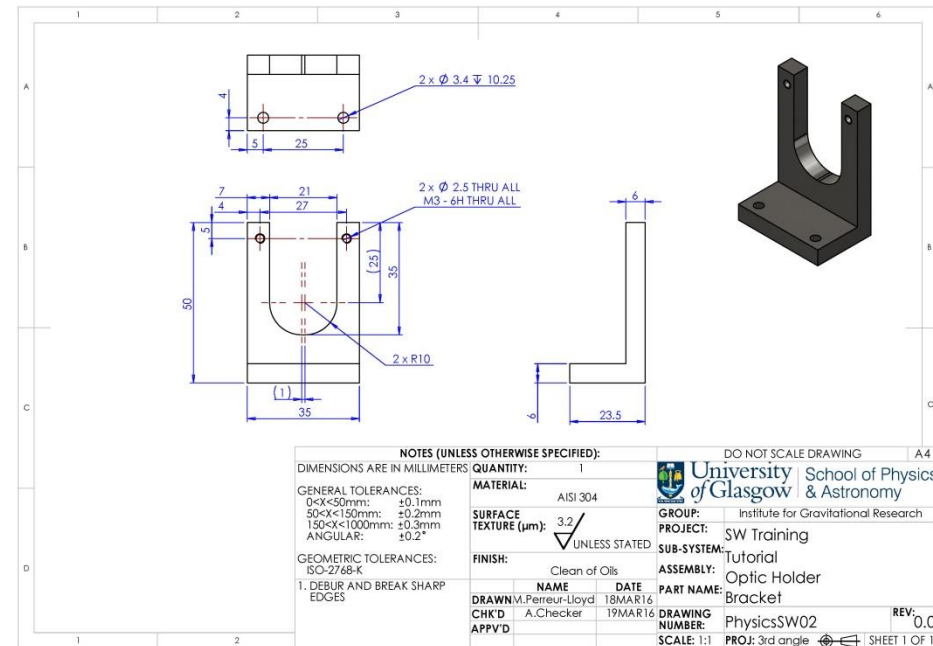
- At a minimum, drawings have at least three orthogonal views and include a isometric view as this helps the manufacturer visualise the part.
- Never dimension to hidden detail
 - use a section view or a detail view
- For larger or more complex parts use a bigger paper size and drawing template – it can be too hard to see detail if it is scaled to fit on A4 or A3!
- UK & US drawings typically use 3rd point perspective. Continental European drawings may be drawn in 1st point perspective.
- Older drawing standards, derived from when it was all done with pencil and paper, require that drawings are done in black and white (using thick and thin lines). Now though, from my point of view, there is no harm in using a bit of colour to make things clearer!



Engineering Drawings (2)

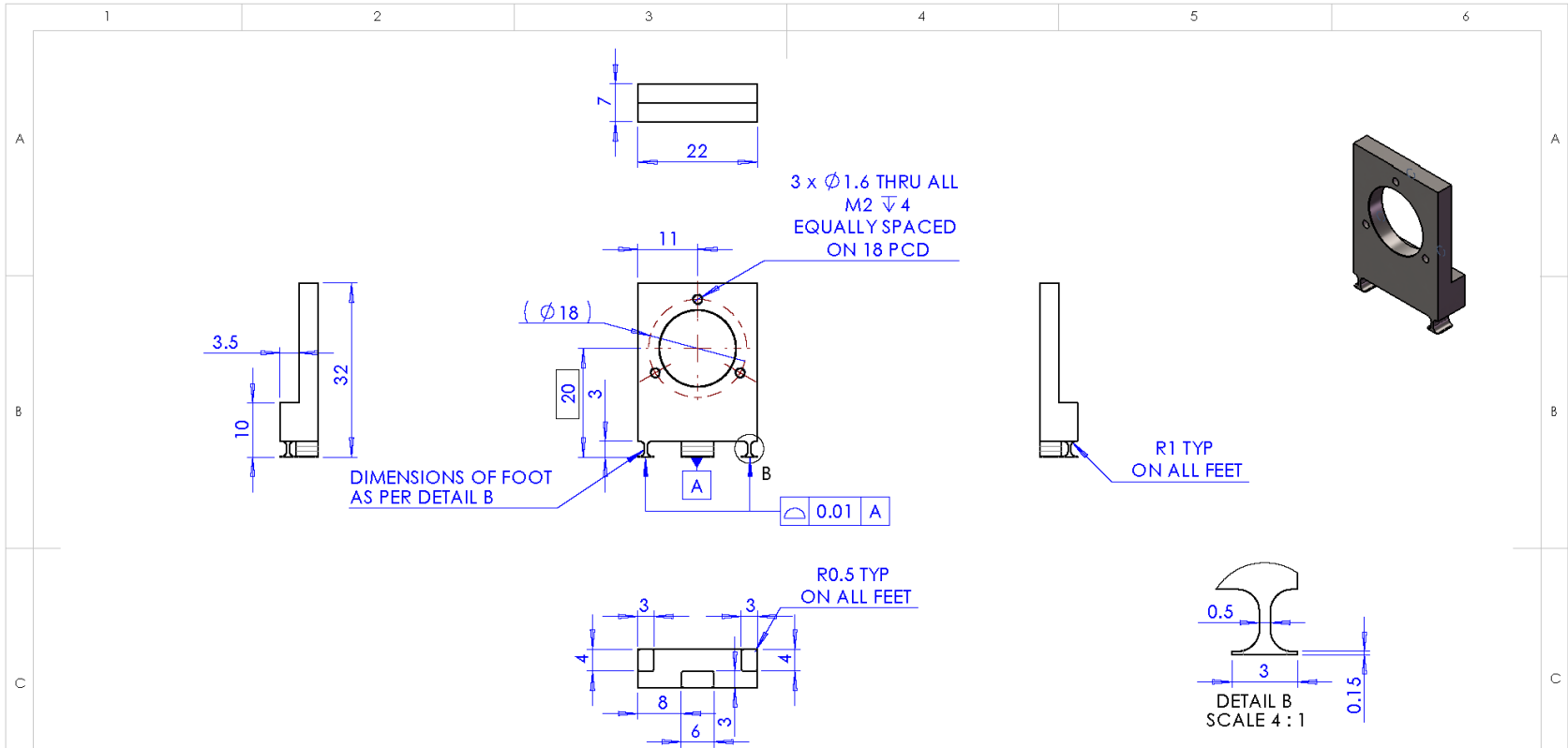
General rules continued...

- Do not put tight tolerances on the part where they are not required. This will only add to the cost of manufacture.
- Only include dimensions that are required. Only the dimensions (and their associated tolerances) shown on the engineering drawing are used for manufacture.
 - i.e. if a 50mm wide part is symmetrical in the middle, it is not necessary to include two 25mm dimensions from a centreline to the edge of the part as well as a 50mm dimension of the full width of the part – the machinist then does not know which dimension to take the tolerance from.



- The title area in the bottom right hand corner is where all additional manufacturing notes should be contained. The majority of this is filled automatically using the customised tools, but additional notes may be added.

Engineering Drawings Examples



NOTES (UNLESS OTHERWISE SPECIFIED):

DIMENSIONS ARE IN MILLIMETERS


GENERAL TOLERANCES:
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GEOMETRIC TOLERANCES:
 ISO-2768-K

1. DEBUR AND BREAK SHARP EDGES
2. 3D CAD MODEL OF PART TO BE SUPPLIED

QUANTITY: 5

MATERIAL:
 Titanium (ASTM Grade 2)

SURFACE TEXTURE (μm): 3.2
 UNLESS STATED

FINISH:
 Clean of Oils

NAME	DATE
DRAWN M.Perreux-Lloyd	02MAR11

CHK'D
 APPV'D

DO NOT SCALE DRAWING

A4



Department of Physics and Astronomy
 Institute for Gravitational Research

PROJECT: LISA

SUB-SYSTEM: EBB

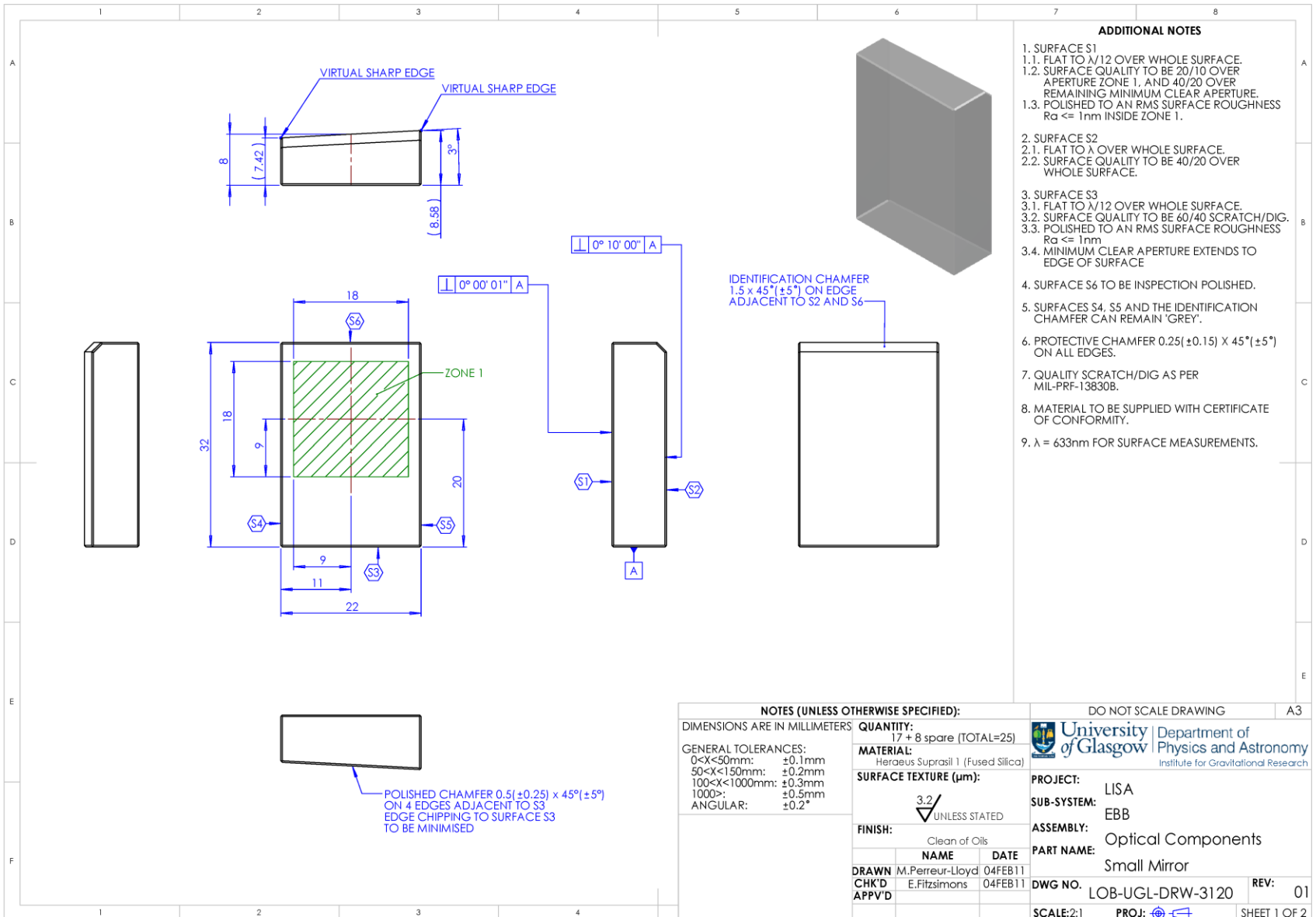
ASSEMBLY: Small Waveplate Holder

PART NAME: Small Waveplate Bracket

DWG NO. LOB-UGL-DRW-3077 **REV:** 00

SCALE: 1:1 **PROJ:**  SHEET 1 OF 1

Engineering Drawing Examples (2)

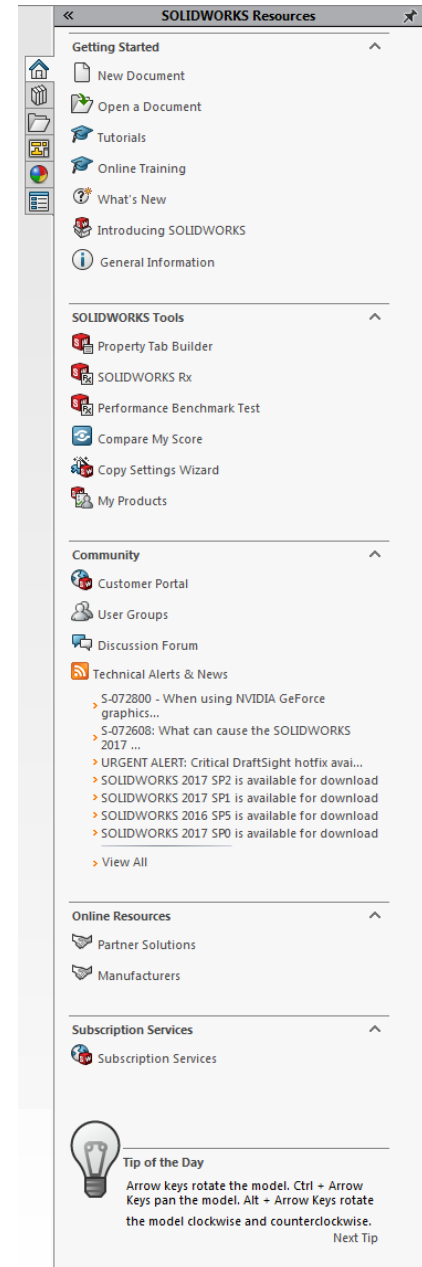


Other Resources

- A number of SolidWorks Resources are available from the right hand panel in the SolidWorks GUI.
 - Online Training Tutorials
 - 3D Content Central
 - available through the design library tab
 - access to a library of standard parts drawn up by companies or other SolidWorks users
- www.solidworks.com
- Russell Jones and I (ext. 3376) are happy to assist you with any problems that you are having using SolidWorks.

Now continue to:

Tutorial Four - Engineering Drawings



SOLIDWORKS Resources

Getting Started

- New Document
- Open a Document
- Tutorials
- Online Training
- What's New
- Introducing SOLIDWORKS
- General Information

SOLIDWORKS Tools

- Property Tab Builder
- SOLIDWORKS Rx
- Performance Benchmark Test
- Compare My Score
- Copy Settings Wizard
- My Products

Community

- Customer Portal
- User Groups
- Discussion Forum
- Technical Alerts & News
 - 5-072800 - When using NVIDIA GeForce graphics...
 - 5-072608: What can cause the SOLIDWORKS 2017 ...
 - URGENT ALERT: Critical DraftSight hotfix avail...
 - SOLIDWORKS 2017 SP2 is available for download
 - SOLIDWORKS 2017 SP1 is available for download
 - SOLIDWORKS 2016 SP5 is available for download
 - SOLIDWORKS 2017 SP0 is available for download
 - View All

Online Resources

- Partner Solutions
- Manufacturers

Subscription Services

- Subscription Services

Tip of the Day

Arrow keys rotate the model, Ctrl + Arrow Keys pan the model, Alt + Arrow Keys rotate the model clockwise and counterclockwise.
Next Tip