

Worksheet – Introduction COSMOSWorks.

This is a quick introduction to using COSMOSWorks (which is a SolidWorks ‘add in’ package).

COSMOSWorks is a Finite Element Analysis (FEA) package. We will learn here how to use it to simulate the elastic behaviour of engineered objects and then analyse the resultant stresses.

On line documents: www.plymouth.ac.uk/cad

1.0 SolidWorks modelling again:

For a quick introduction to SolidWorks work through these online tutorials:

Help > Online tutorial

Online tutorial title:	Completed:
1) 30 minute lesson.	
2) Lesson 1 – Parts	
3) Lesson 2 – Assemblies	
4) Assembly mates	
5) Fillets	
6) Lofts	
7) Pattern Features	
8) Revolves & Sweeps	

2.0 Basic introduction to COSMOSWorks.

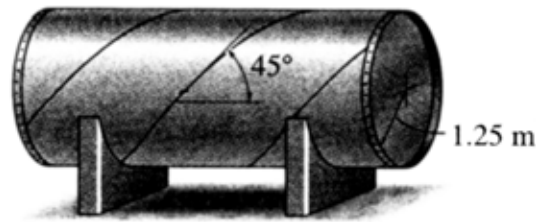
Work through Lesson 1 of the: COSMOS Student Workbook:
Basic Functionality of COSMOSWorks.

Here you will perform a static analysis on a SolidWorks assembly.

3.0 A practical application - using shells and symmetry.

Here you will use COSMOSWorks to help you solve the problem presented in:
Statics and mechanics of materials by R.C. Hibbeler.

15-38. The cylindrical pressure vessel has an inner radius of 1.25 m and a wall thickness of 15 mm. It is made from steel plates that are welded along the 45° seam. Determine the normal and shear stress components along this seam if the vessel is subjected to an internal pressure of 8 MPa.



Prob. 15–38

Create a model in SolidWorks of the pressure vessel detailed in the above problem. We will assume gravity loads are not to be considered so that we can make use of symmetry. Only model 1/8 of the vessel. Make the model using surfaces, not solids. What do you think the length of the model should be?

Then follow the steps below, as you did in section 2.0:

Step 1: Create a study

Under name, enter your choice of name and select **Shell mesh using surfaces**. Under type select **Static**. Select the **Large displacement** flag.

Step 2: Assign material

Plain carbon steel.

Step 3: Apply restraints

This particular example, using surfaces, is tricky, as the **Symmetry** restraints option can only be applied to surfaces of solids. Use the **Use reference geometry** option to build up your own symmetrical restraints for each of the edges.

Step 4: Apply loads

This would be a **Pressure** type, **Normal to selected surface**, 8MPa, or 8 000 000 N/m².

Step 5: Mesh the model

Use the default size parameters.

Step 6: Run the analysis

Step 7: Visualise the results

Right click the default **Plot1 – (vonMises)** result set and select **Show**.

Animate the results first, just to confirm that the restraints have worked as expected:

COSMOSWorks > Result tools > Animate

Extract the longitudinal and hoop stresses from our analysis model, in order to calculate the normal and shear stresses along the welded seam. We will probe the stress values along model edges, resolved relative to a selected plane.

COSMOSWorks > Plot results > Stress...

Create a plot of the appropriate **Normal stress** relative to an appropriately selected reference plane. **Show** the plot and then right click **Probe**. Click at points along the edge to find stress values.

Alternatively select **List selected** and click the edge to be examined, click **Update** and **Plot**. This shows how the stress varies along the edge. Move pointer over plot curve to display stress values.

3.0 Further work.

Browse to:

C:\Program Files\SolidWorks\COSMOS\Help

Open the **VCWLinear.chm** file. This is a help chapter which provides a wide selection of classic analysis problems which can be used to verify analysis results. Work through any of these which suit the analyses you wish to carry out for your project/design work.