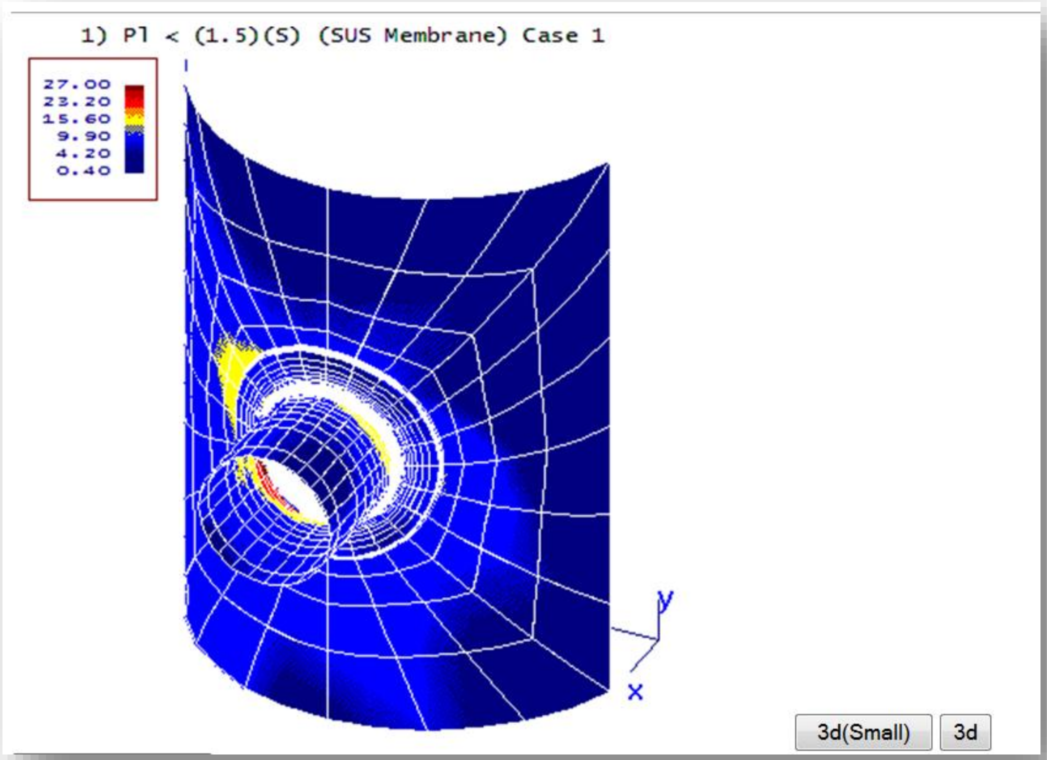
CAU2014

Using WRC 107 and NozzlePRO FEA



Presented by: Ray Delaforce

INTERGRAPH

History of WRC 107

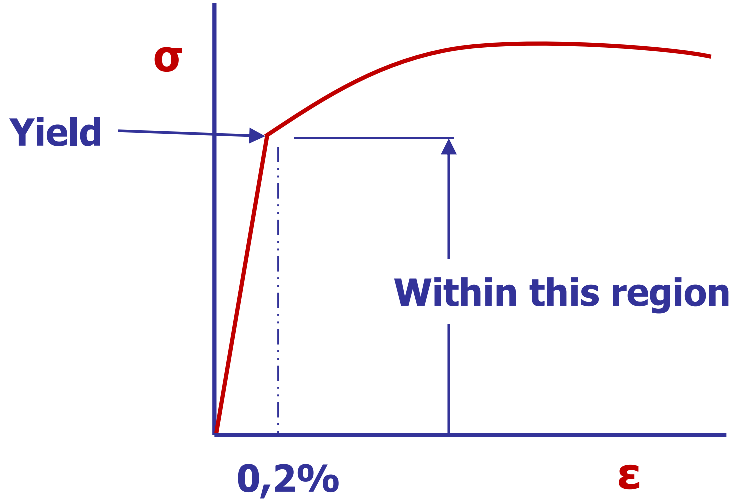
Pressure vessel analysis was handled by the codes, such as:

- ❑ ASME Section VIII, Division 1
- ❑ ASME Section VIII, Division 2
- ❑ PD 5500 (British Code)
- ❑ EN 13445 (European Code)
- ❑ AD 2000 (Merkblatter) (German Code)
- ❑ CODAP (French Code)

They mainly concentrated on primary membrane stresses (more later)

These stresses had to be below the yield stress of the metal

The strain is not to exceed about 0,2%



History of WRC 107

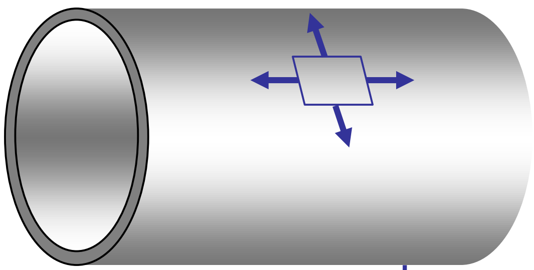
Example of membrane stress in a cylinder subject to internal pressure

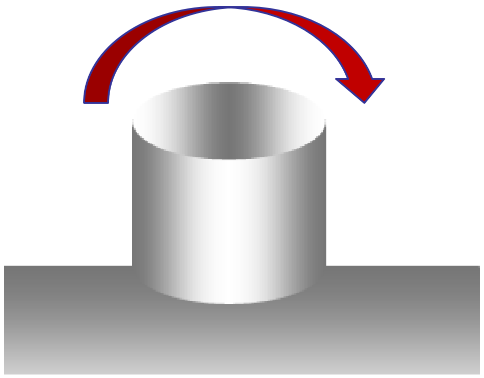
The code allowable stress is about here, below yield

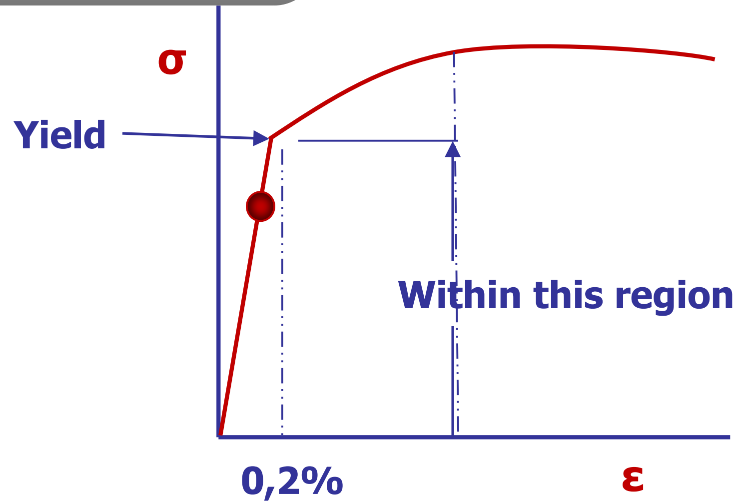
In the case of nozzles subject to external loads, the stresses can be here

That is beyond the scope of the codes

Strains go beyond 0,2%







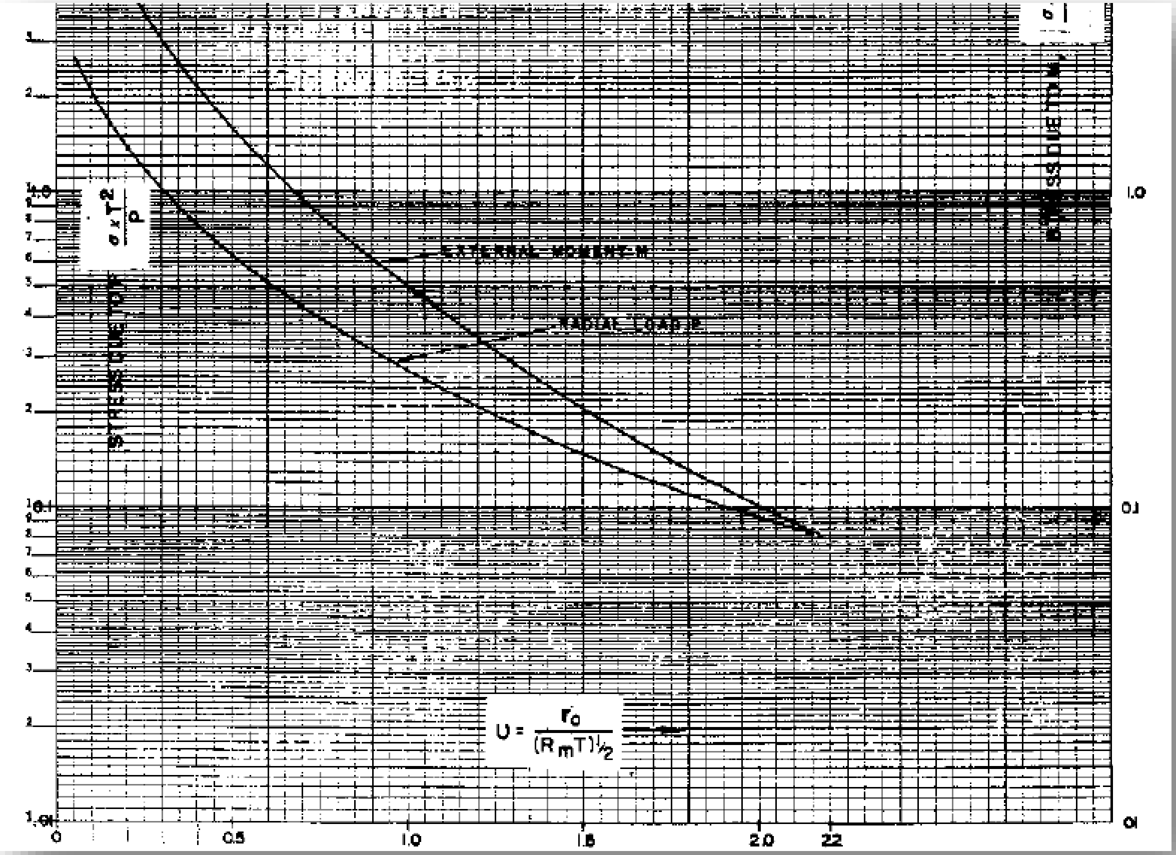
History of WRC 107

Stress analysis is really required

WRC107 was published for this purpose

WRC107 offered a graphical method of doing the stress analysis

Here is a typical graph:



History of WRC 107

Stress analysis is really required

WRC107 was published for this purpose

WRC107 offered a graphical method of doing the stress analysis

Here is a typical graph:

Many of the graphs are difficult to read – more of that later

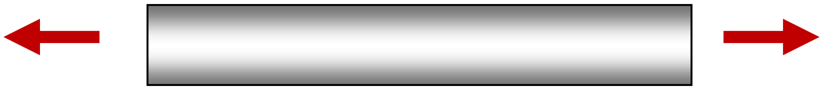
WRC107 first published in 1965

- Revised 1968
- Revised 1970
- Revised 1972
- Revised 1979

Based upon the theoretical work of Prof. Biljaard of Cornell University
Attachments (nozzles) on cylinders and spheres only can be analysed
There are also geometric limitations
Using WRC 107 is very tedious and time consuming
PV Elite and CodeCalc make this analysis simple and fast

Let us look at some principles of stress analysis

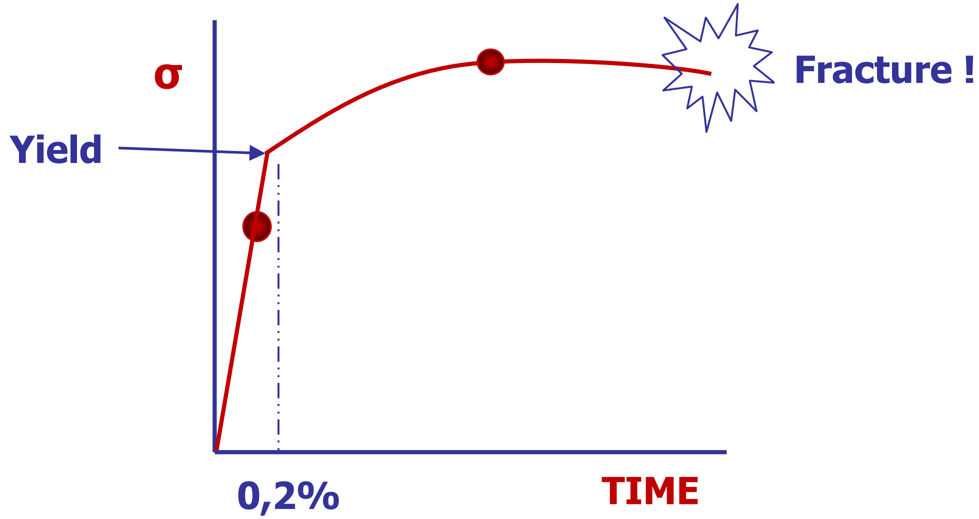
Consider a bar subject to a tensile force



Once the stress passes yield, it continues to stretch until collapse occurs

So, we could re-label the strain axis as time to collapse

This shows we must under normal conditions stay below yield



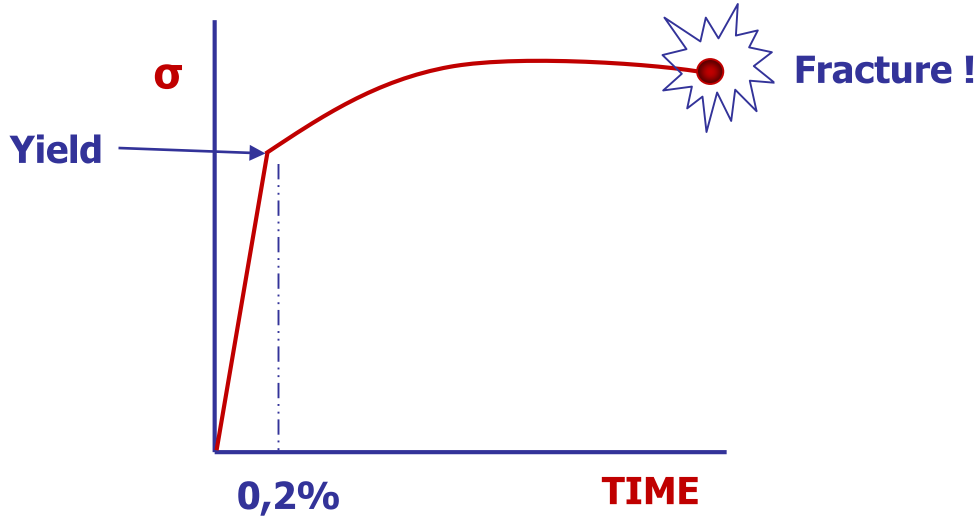
Let us look at some principles of stress analysis

Suppose the force were delivered by this arrangement

If the weight were heavy enough – fracture would occur

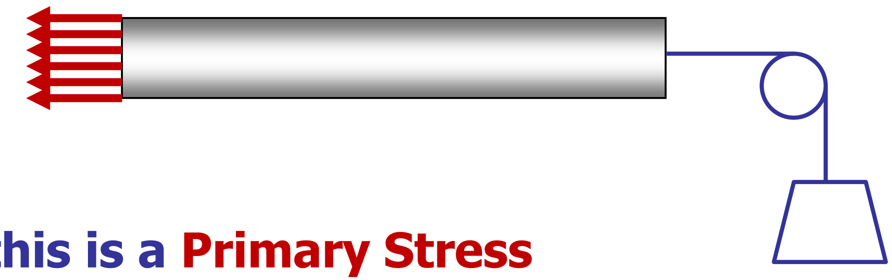
The stress is directly proportional to the load

σ = force / area thus stress is directly related to the load (force)



Let us look at some principles of stress analysis

σ = force / area thus stress is directly related to the load (force)

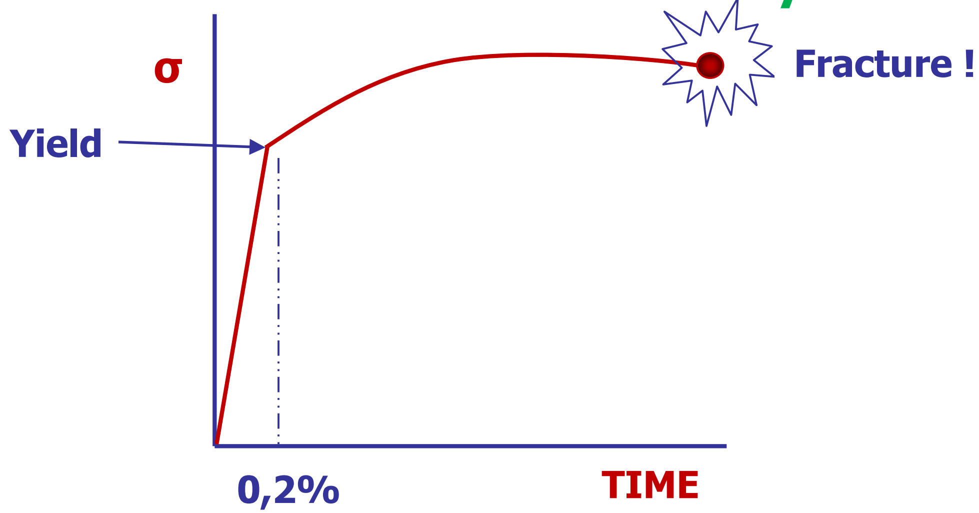


By definition this is a Primary Stress

- The Internal stress is directly related to the external load
- Fracture will occur as the stress increases beyond yield

The stress also exist Everywhere in the bar – membrane stress

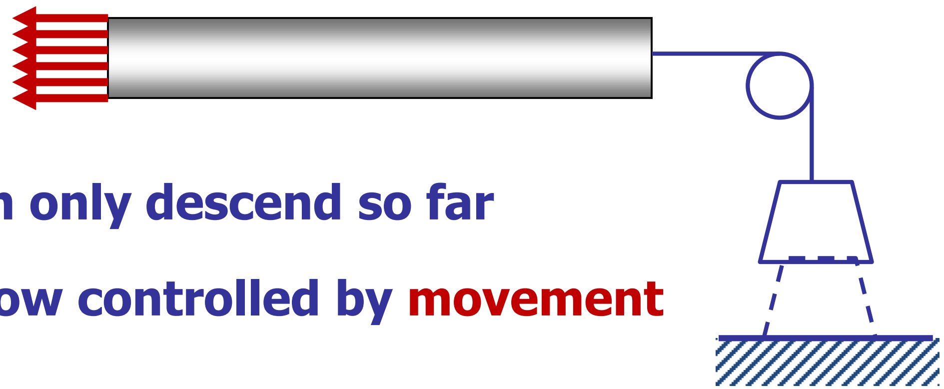
- Everywhere designates the stress as a General Stress
- Therefore we have a General Primary Membrane stress



INTERGRAPH

Let us look at some principles of **stress analysis**

Suppose now we have table under the weight to restrict its movement

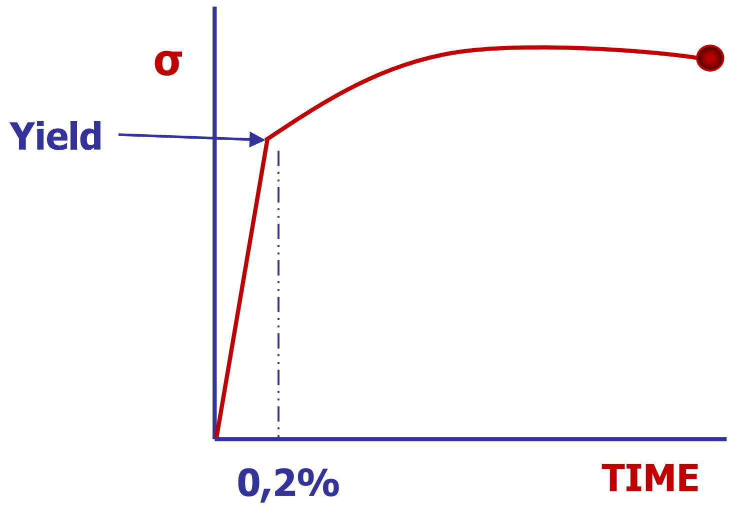


The weight can only descend so far

The stress is now controlled by **movement**

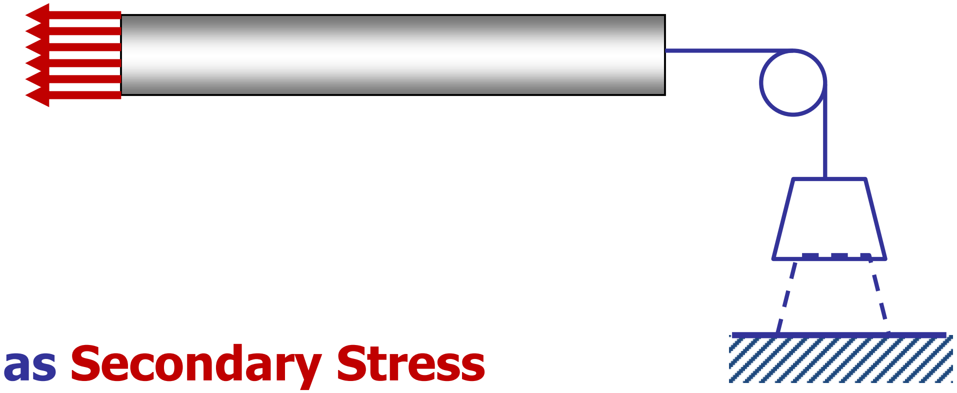
Stress does not reach fracture, because movement is limited

The weight is **NOT** related to the stress. Only to the **movement**



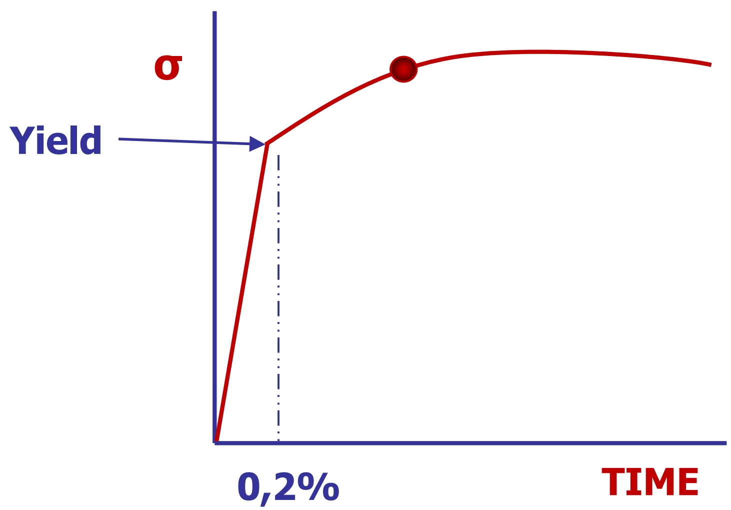
Let us look at some principles of stress analysis

The weight is NOT related to the stress. Only to the movement



This is known as Secondary Stress

- Controlled by movement of another element
- Internal stress not related to the force in the other component
- Also called strain related

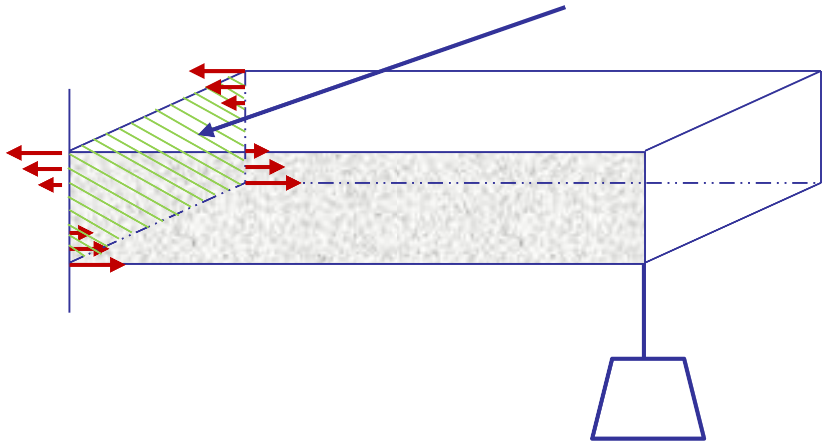


Let us look at some principles of stress analysis

Now we consider a Cantilever subject to un-restricted force

There is an internal bending stress in the beam

This bending stress can exist everywhere at this location



This stress has the following characteristics

- **The internal stress is directly related to the load - Primary**
- **It exists Everywhere –General**

This is a General Primary Bending stress

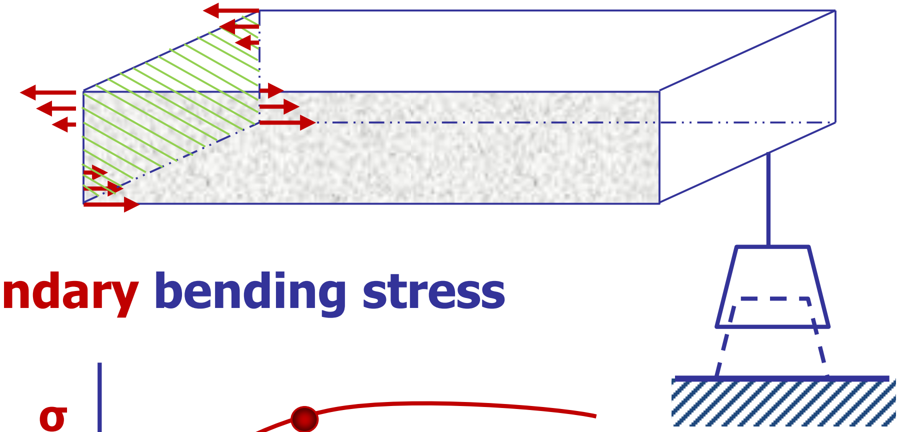
Let us look at some principles of stress analysis

We again restrict the motion of the weight

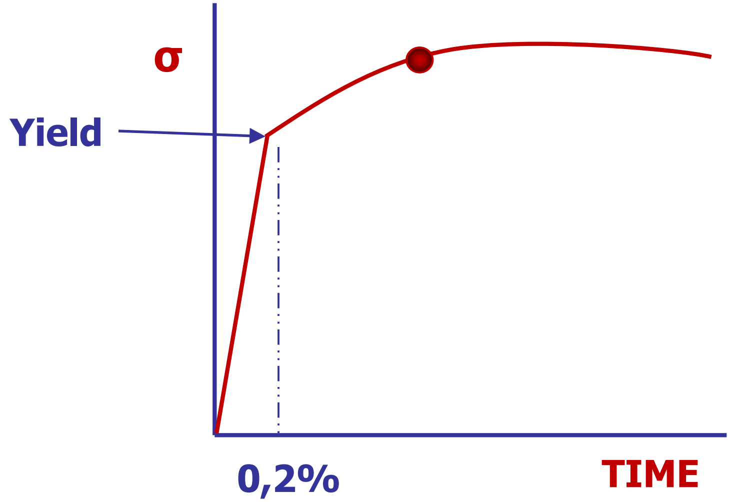
The bending stress is NOT directly related to the load (weight)

The bending strain is restricted, and does not proceed to fracture

It is strain controlled, and not load controlled



We have a Secondary bending stress



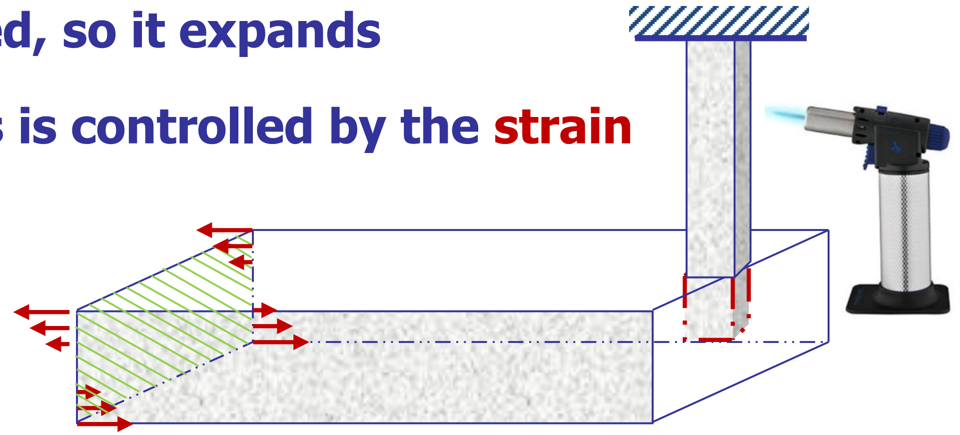
INTERGRAPH

Let us look at some principles of **stress analysis**

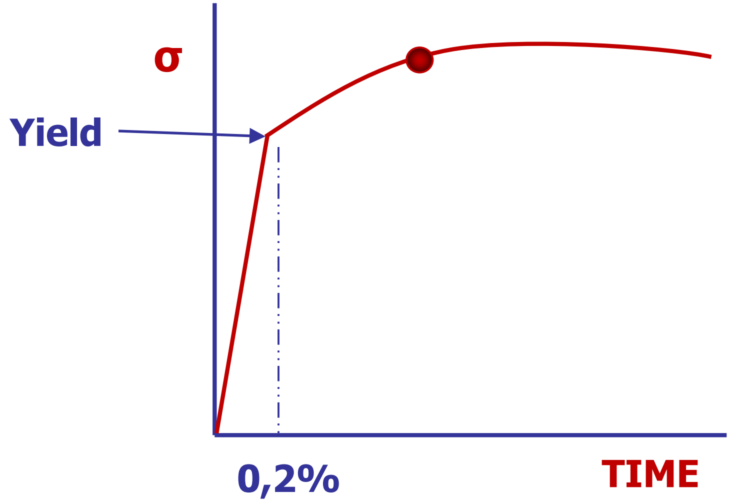
Now look at this arrangement

The rod is heated, so it expands

Again the stress is controlled by the **strain**



This is also a **Secondary stress** – also called an **Expansion stress**



Let us look at some principles of stress analysis

Sum up what we have learned so far:

Primary stress	**Secondary stress**
Directly related to the imposed load: eg $\sigma = F/A$	Relate to the strain induced in the component – strain controlled
If the load is great, failure can occur	The strain is restricted so failure does not occur
	Often from expansion of another component – example, thermal expansion

There is one more stress category we have deal with

INTERGRAPH

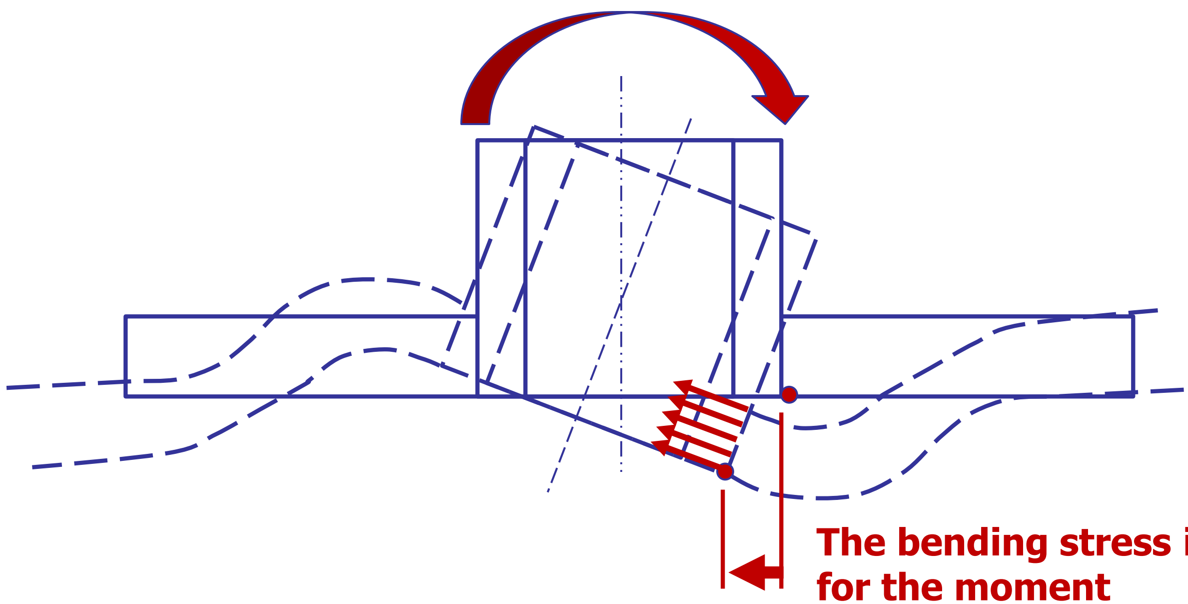
Let us look at some principles of stress analysis

Consider a nozzle in a shell subject to an external moment

The shell deforms to accommodate the moment like this

Notice how it pulls the shell to the left, giving rise to a membrane stress

This stress fades away rapidly from the nozzle to shell junction



INTERGRAPH

Let us look at some principles of **stress analysis**

Consider a nozzle in a shell subject to an external moment

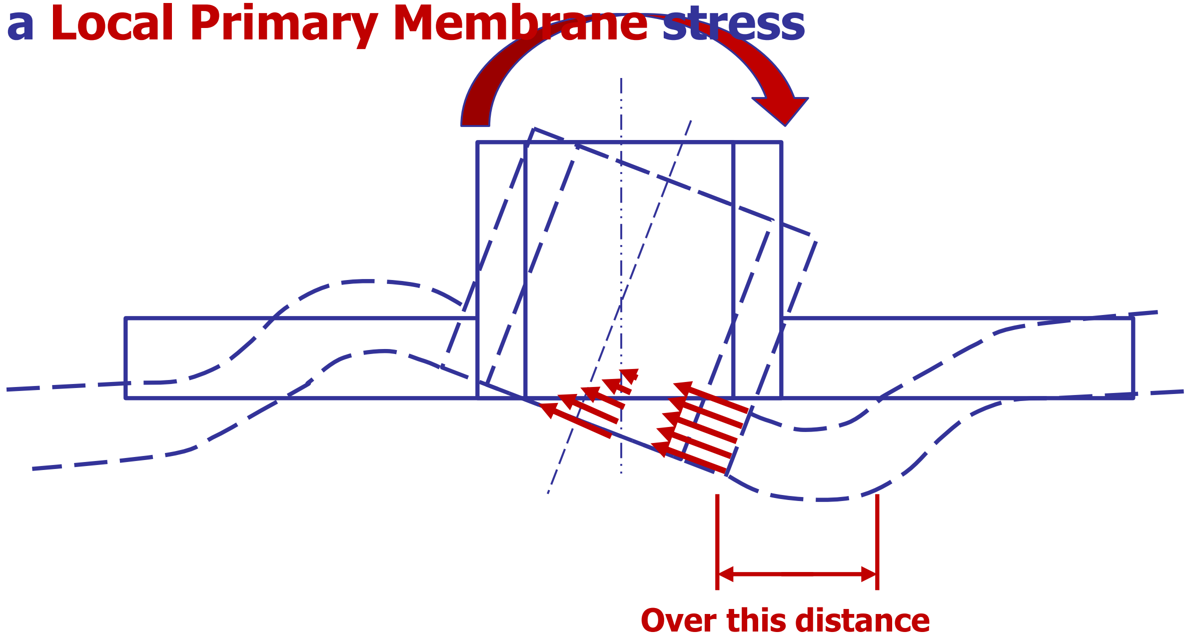
The shell deforms to accommodate the moment like this

Notice how it pulls the shell to the left, giving rise to a membrane stress

This stress **fades away** rapidly from the nozzle to shell junction

That stress does **not** exist everywhere, therefore it is **LOCAL**

This is a **Local Primary Membrane stress**



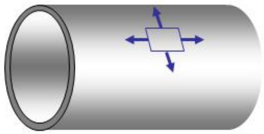
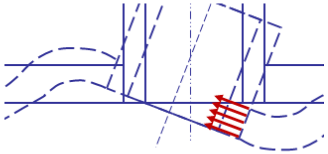
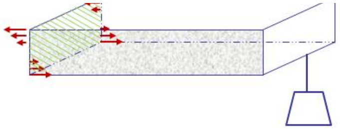
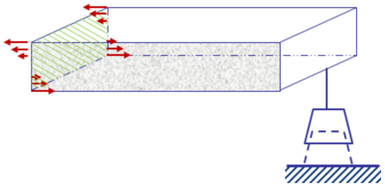
The bending stress is treated as **Secondary** stress

Let us look at some principles of stress analysis

Summing what we have learned so far:

ASME Section VIII, Division 2 gives them symbols

These stresses can exist in combination

Stress type	Symbol	Allowable stress	
General Primary Membrane	P_m	S	
Local Primary Membrane	P_L	1,5S	
Primary Bending	P_b	1,5S	
Secondary: ❑ Local Membrane ❑ Local Bending	Q	3S or $2S_y$	

Let us look at some principles of stress analysis

Summing what we have learned so far:

ASME Section VIII, Division 2 gives them symbols

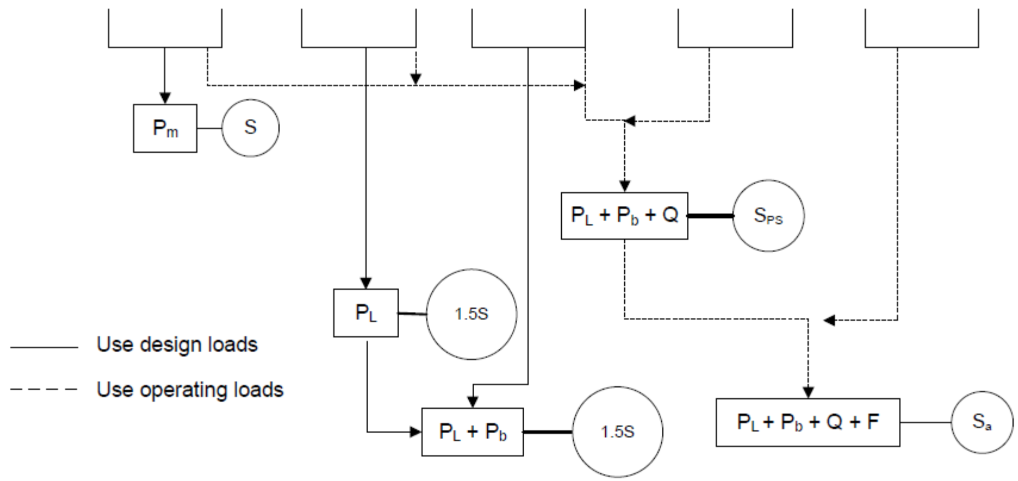
These stresses can exist in combination

Stress type	Symbol	Allowable stress
Primary	P_m	S
Primary + Local	$P_m + P_L$	1,5S
Primary + Local	$P_m + P_L + P_B$	1,5S
Primary + Local + Secondary	$P_m + P_L + P_B + Q$	3S of $2S_y$

Let us look at some principles of stress analysis

This is the ‘Hopper’ diagram from ASME Section VIII, Division 2

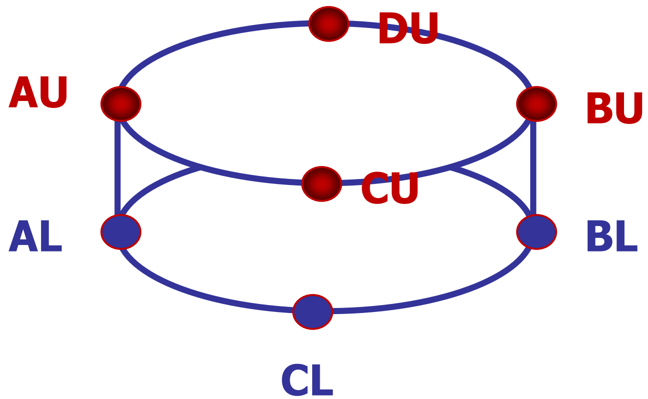
Stress Category	Primary			Secondary Membrane plus Bending	Peak
	General Membrane	Local Membrane	Bending		
Description (For examples, see Table 5.2)	Average primary stress across solid section. Excludes discontinuities and concentrations. Produced only by mechanical loads.	Average stress across any solid section. Considers discontinuities but not concentrations. Produced only by mechanical loads.	Component of primary stress proportional to distance from centroid of solid section. Excludes discontinuities and concentrations. Produced only by mechanical loads.	Self-equilibrating stress necessary to satisfy continuity of structure. Occurs at structural discontinuities. Can be caused by mechanical load or by differential thermal expansion. Excludes local stress concentrations.	1. Increment added to primary or secondary stress by a concentration (notch). 2. Certain thermal stresses which may cause fatigue but not distortion of vessel shape.
Symbol	P_m	P_L	P_b	Q	F



WRC 107 Force and Moment Convention

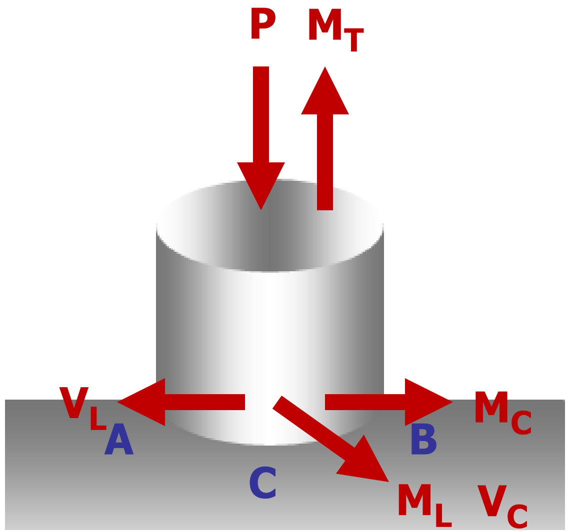
Nozzle removed – here is the hole

Labeling convention



U = upper surface (outside)

L = lower surface (inside)



Radial force	P
Longitudinal force	V_L
Circumferential force	V_C
Longitudinal moment	M_L
Circumferential moment	M_C
Torsional moment	M_T

Type of Loading per normal stress analysis

Sustained load

- Loads that are there for long periods
- Pressure
- Weight (on an attachments)

Occasional

- Loads that are momentary (lasting a short time)
- Wind loading
- Seismic loading
- Because they are short lived – add 20% to allowable stress

Expansion

- Strain controlled
- From thermal expansion of piping
- Always Secondary Stresses

Operating loads

- These are commonly Sustained + Thermal (eg: CAESAR II)

WRC 107 Demonstration (First)

Here are the regions around the nozzle

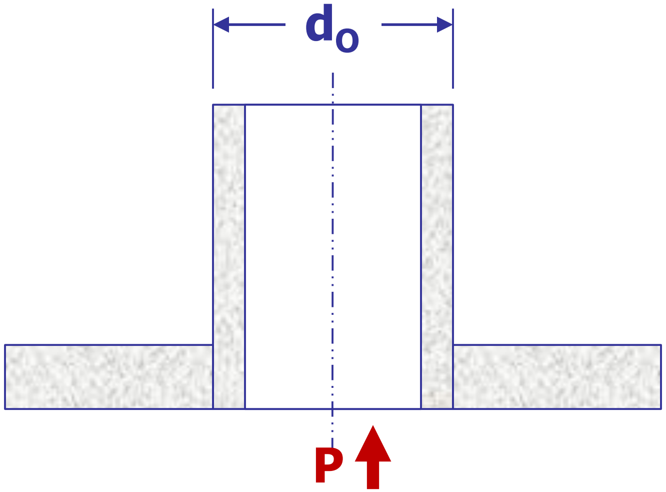
Stresses in the Vessel at the Attachment Junction

Type of Stress	Load	Au	Al	Bu	Bl	Cu	Cl	Du	Dl
		Stress Values at (MPa)							
Circ. Memb.	P	-24	-24	-24	-24	-19	-19	-19	-19
Circ. Bend.	P	-58	58	-58	58	-90	90	-90	90
Circ. Memb.	MC	0	0	0	0	-7	-7	7	7
Circ. Bend.	MC	0	0	0	0	-107	107	107	-107
Circ. Memb.	ML	-23	-23	23	23	0	0	0	0
Circ. Bend.	ML	-46	46	46	-46	0	0	0	0

Here are the final stress final stress Categories

Type of Stress Int.	Max. S.I. MPa	S.I. Allowable		Result	
Pm (SUS)	59.41	137.90	S	Passed	
Pm (SUS+OCC)	59.41	165.48	1,2S	Passed	
Pm+Pl (SUS)	101.80	206.85	1,5S	Passed	
Pm+Pl (SUS+OCC)	100.65	248.22	1,8S	Passed	1,8=1,2 x 1,5
Pm+Pl+Q (TOTAL)	216.15	413.70	3S	Passed	

WRC 107 Does not discuss **Pressure Thrust** on the nozzle



Thrust $F = P\pi d_o^2/4$**, this can be added in PV Elite (Demo)**

Before

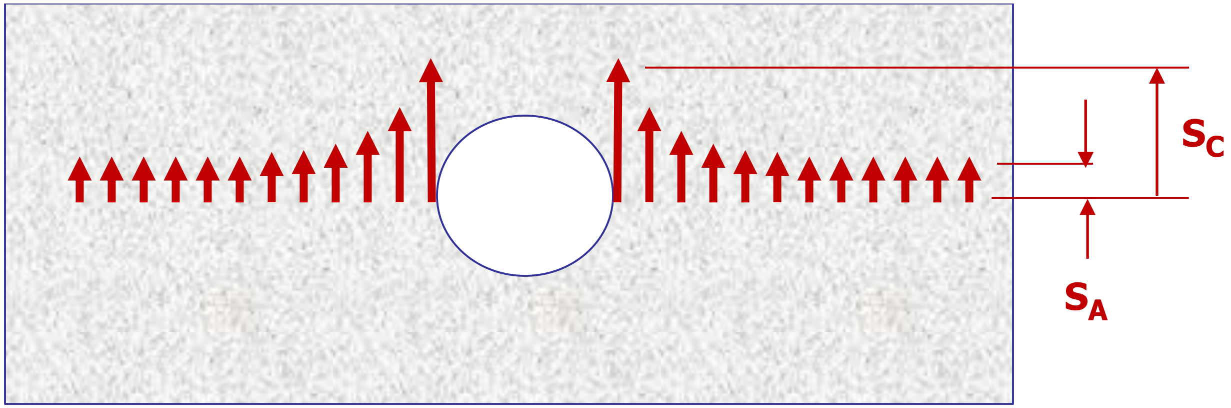
Type of Stress Int.	Max. S.I. MPa
Pm (SUS)	94.38
Pm (SUS+OCC)	94.38
Pm+Pl (SUS)	93.82
Pm+Pl (SUS+OCC)	84.03
Pm+Pl+Q (TOTAL)	349.95

After

Type of Stress Int.	Max. S.I. MPa
Pm (SUS)	94.38
Pm (SUS+OCC)	94.38
Pm+Pl (SUS)	128.14
Pm+Pl (SUS+OCC)	118.34
Pm+Pl+Q (TOTAL)	253.52

There is a stress concentration at the Nozzle to Shell junction

The stress increases at the junction like this:



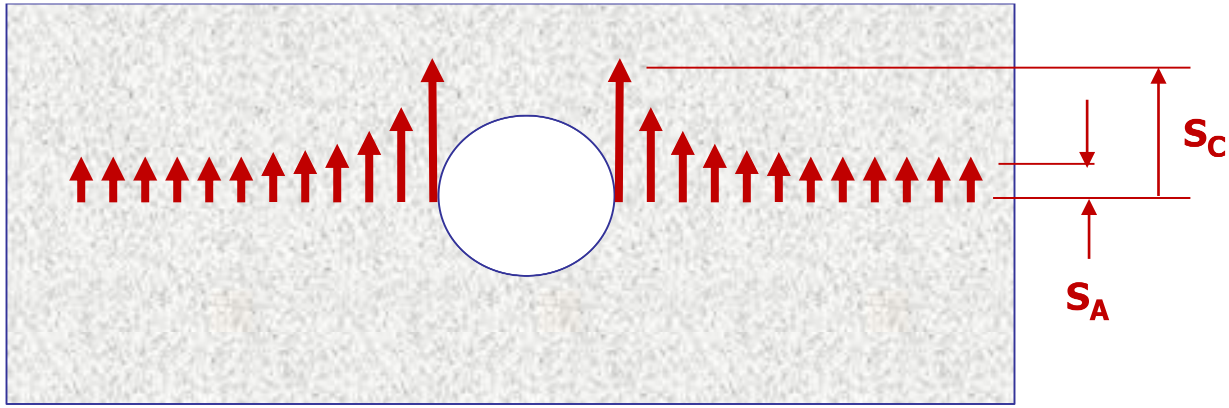
S_A is the average stress, S_C is the increases stress

S_C /S_A is known as the stress concentration factor (scf)

It is usually about 3

ASME VIII, Division 2 calls the scf by the name of Pressure Index

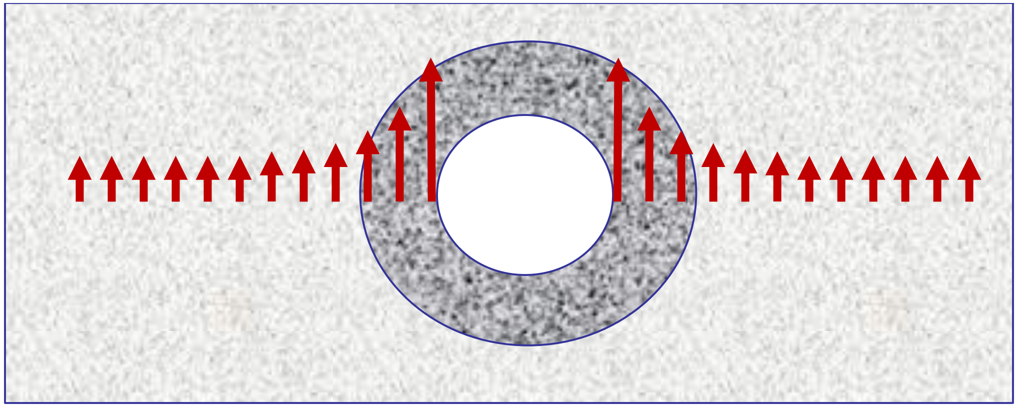
Adding the scf (Pressure Indices) - (Demo)



It now **FAILS**, we can fix the problem be adding a re-pad (Demo)

Type of Stress Int.	Max. S.I. MPa	S.I. Allowable	Result
Pm (SUS)	*301.89*	*137.97*	*Failed*
Pm (SUS+OCC)	*301.89*	*165.56*	*Failed*
Pm+Pl (SUS)	*326.16*	*206.95*	*Failed*
Pm+Pl (SUS+OCC)	*318.45*	*248.34*	*Failed*
Pm+Pl+Q (TOTAL)	348.96	413.91	Passed

We now have two analyses



Nozzle junction

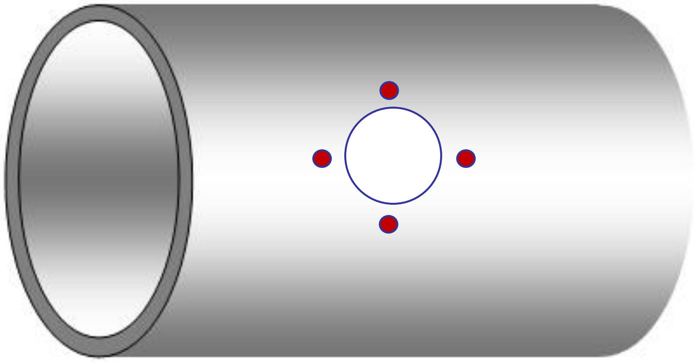
Type of Stress Int.	Max. S.I. MPa
Pm (SUS)	*151.90*
Pm (SUS+OCC)	151.90
Pm+Pl (SUS)	155.85
Pm+Pl (SUS+OCC)	155.55
Pm+Pl+Q (TOTAL)	171.23

Edge of the pad

Type of Stress Int.	Max. S.I. MPa
Pm (SUS)	94.38
Pm (SUS+OCC)	94.38
Pm+Pl (SUS)	111.59
Pm+Pl (SUS+OCC)	104.08
Pm+Pl+Q (TOTAL)	168.28

WRC107 deficiencies

It only considers **4 points** around the nozzle



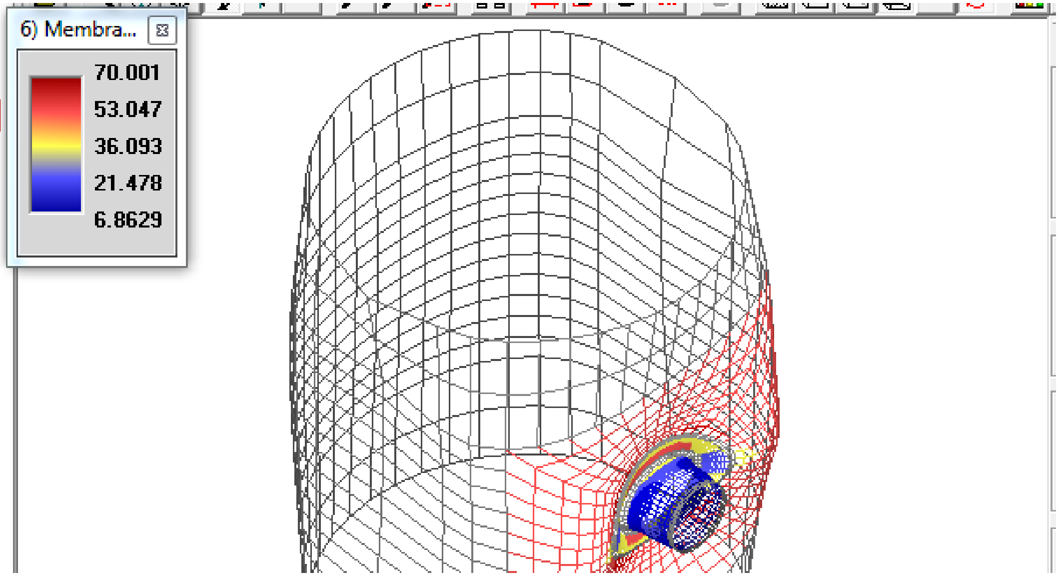
The method is only an approximation

We need better tools to get a better answer

The answer is **FEA !**

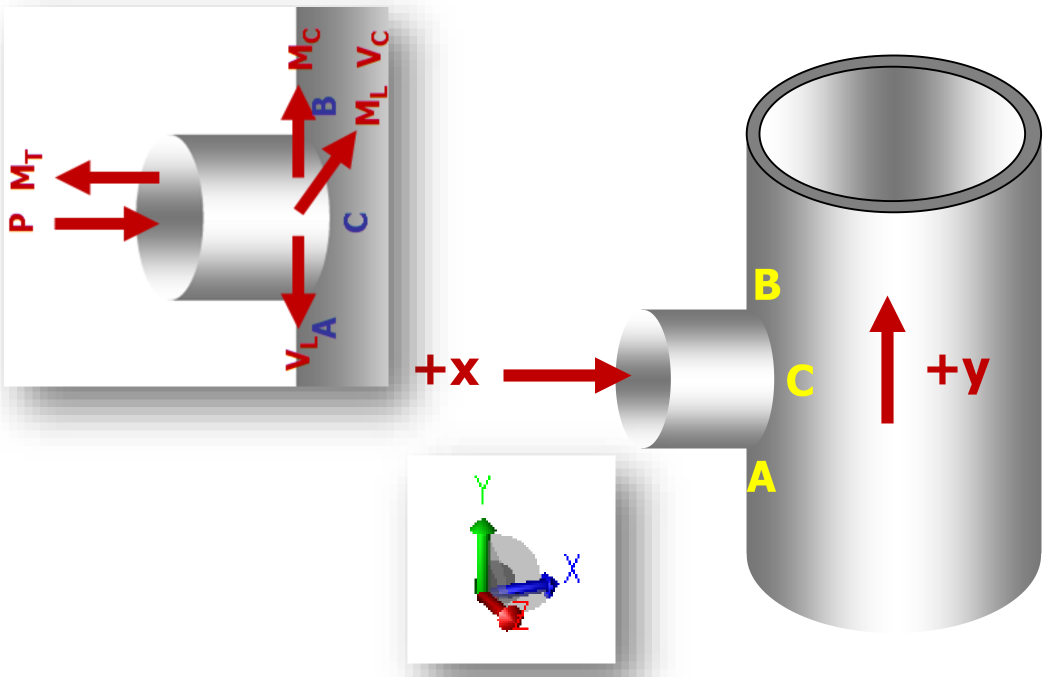
Introduction to NozzlePRO FEA analysis

Now we set the same nozzle up in NozzlePRO (Demo)



Type of Stress Int.	Max. S.I. MPa	S.I. Allowable	Result
Pm (SUS)	95.92	137.90	Passed
Pm (SUS+OCC)	95.92	165.48	Passed
Pm+Pl (SUS)	107.07	206.85	Passed
Pm+Pl (SUS+OCC)	106.70	248.22	Passed
Pm+Pl+Q (TOTAL)	128.19	413.70	Passed

What is a direction cosine ?
Setting the nozzle-vessel in Global Units



Radial force	P
Longitudinal force	V_L
Circumferential force	V_C
Longitudinal moment	M_L
Circumferential moment	M_C
Torsional moment	M_T

Radial force P is in the +X direction

Longitudinal force VL is in the -Y direction from B to A

Circumferential force VC is in the +Z direction from D to C

We need to know the Orientation of the Nozzle and the Vessel

+x and +y are known as Direction Cosines

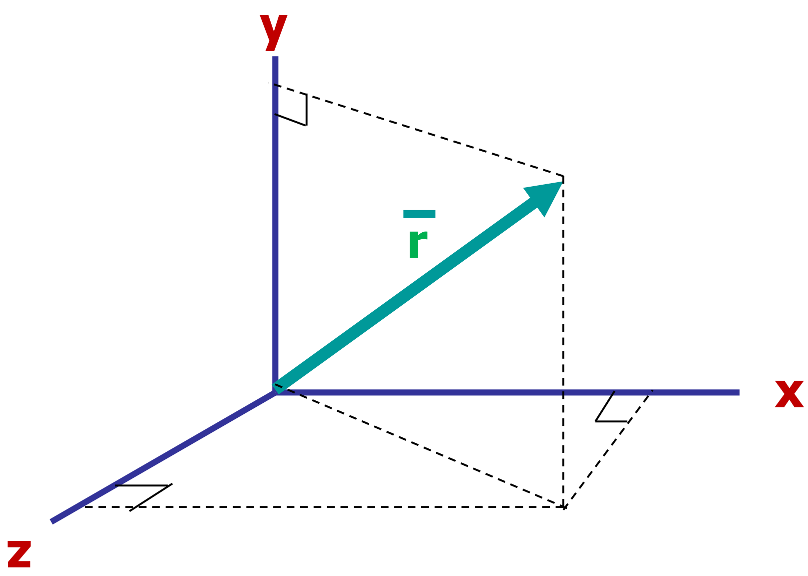
We need to understand Direction Cosines

INTERGRAPH

What is a direction cosine ?

Let us construct a 3 dimensional world, or system

The axes can be thought of as the corners of a box



Label the 3 directions by the letters x, y and z

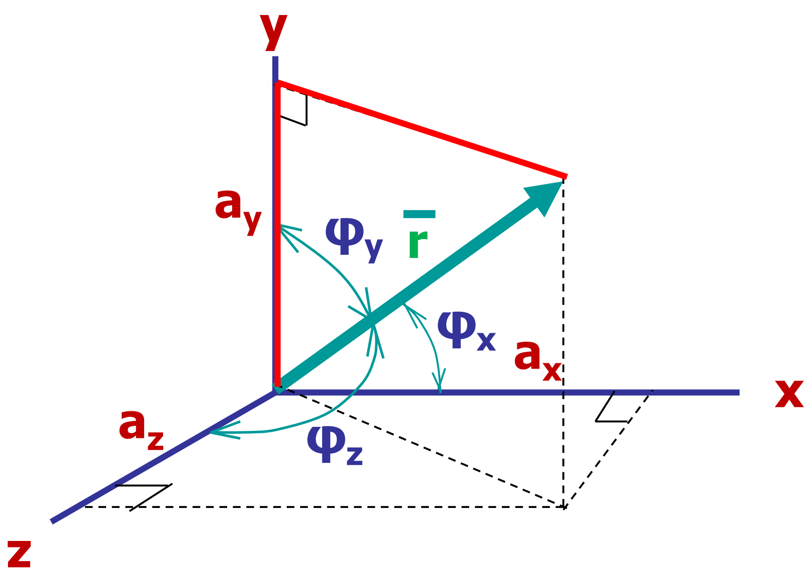
A vector can be represented by an arrow from the origin

The vector can be represented by $\bar{r}$, defines magnitude & direction

The magnitude (length) is defined as $|\bar{r}|$, or simply r

What is a direction cosine ?

These angles define the direction cosines



Now, put in the distances along the axes of the vector

The direction cosine of y is defined as $\cos(\varphi_y) = \dfrac{a_y}{r}$

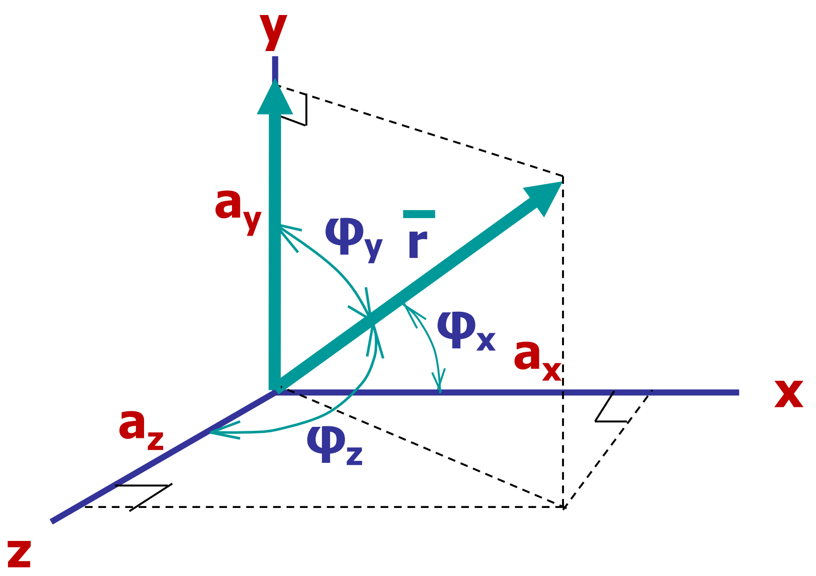
Similarly $\cos(\varphi_x) = \dfrac{a_x}{r}$ **, and** $\cos(\varphi_z) = \dfrac{a_z}{r}$

INTERGRAPH

What is a direction cosine ?

Let the direction cosines be represented by Vx, Vy and Vz

Then $Vx = \cos(\varphi_x)$, $Vy = \cos(\varphi_y)$ and $Vz = \cos(\varphi_z)$



Now, if $Vy = 1$, it follows that $\varphi_y = 0^o$ because $\cos(0^o) = 1$

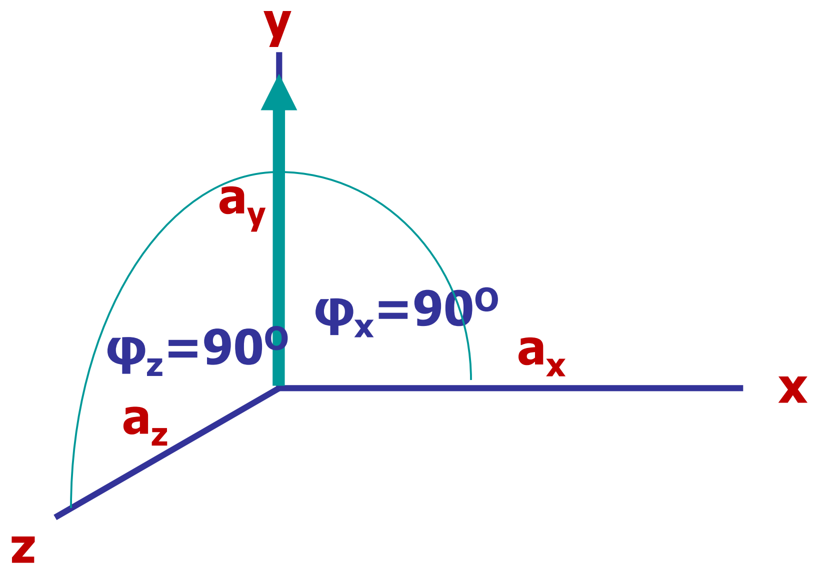
But, if $Vx = 0$, and $Vz = 0$, then $\varphi_x = 90^o$ and $\varphi_z = 90^o$

The vector would then point in the direction of +y

What is a direction cosine ?

Let the direction cosines be represented by Vx, Vy and Vz

Then $Vx = \cos(\Phi_x)$, $Vy = \cos(\Phi_y)$ and $Vz = \cos(\Phi_z)$



Now, if $Vy = 1$, it follows that $\varphi_y = 0^o$ because $\cos(0^o) = 1$

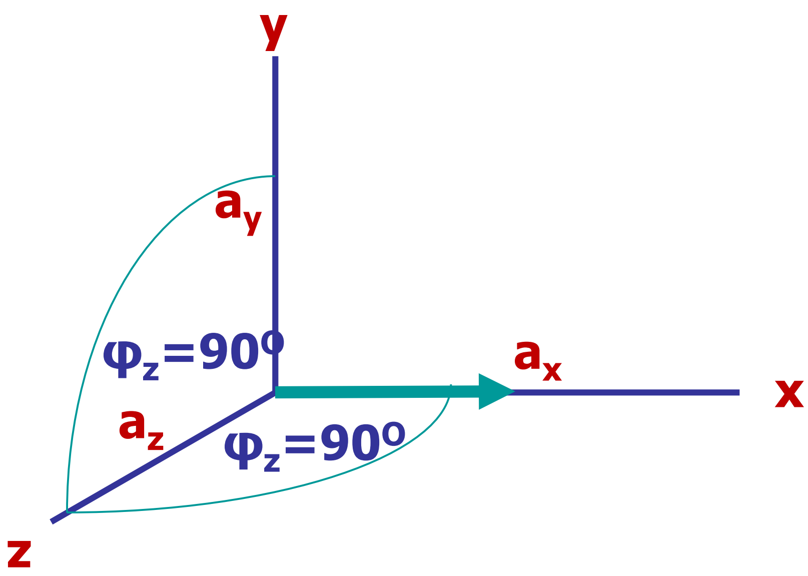
But, if $Vx = 0$, and $Vz = 0$, then $\boxed{\varphi_x = 90^o}$ and $\boxed{\varphi_z = 90^o}$

The vector would then point in the direction of +y

What is a direction cosine ?

Let the direction cosines be represented by Vx, Vy and Vz

Then Vx = $\cos(\Phi_x)$, Vy = $\cos(\Phi_y)$ and Vz = $\cos(\Phi_z)$



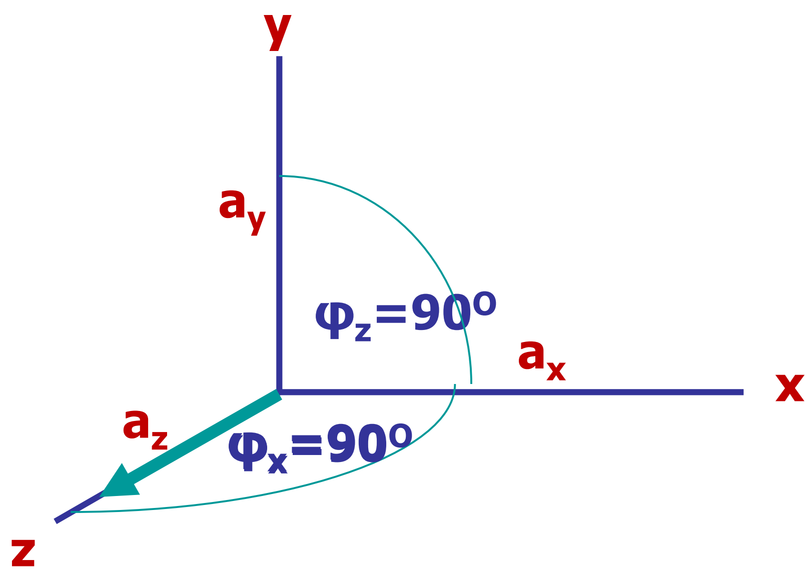
If Vz = 0, Vx = 1 and Vy = 0, the vector would point along x axis

What is a direction cosine ?

Let just direction cosines be represented by Vx, Vy and Vz

We set the direction cosine to 1 for the particular direction

Then Vx = cos(Φ_x) , Vy = cos(Φ_y) and Vz = cos(Φ_z)



If Vx = 0, Vy = 0 and Vz = 1, the vector would point along z axis

What is a direction cosine ?

We just set the direction cosine to 1 for the particular direction