### **Overview**

Silo

### 3-D Linear Static Analysis

### Model

- Unit : N, m
- Isotropic Elastic Material
- Plate Elements

### Load & Boundary Condition

- Self Weight
- Face Pressure
- Constraint

### Result Evaluation

- Deformation
- Maximum Shear Stress
- at Element Mid-Plane







- Model Type : [3D] 1
- Unit System : [N, m] 2
- Click [OK] Button 3
- Click [Move Work Plane] 4
- Select [XZ-Plane] 5
- Click [OK] Button 6
- Click [Normal View] 7



 $\times$ 

0

0,0,0

Apply

Cancel









- Select [1D →2D] tab
- 2 Select [Geometry Edge→2D Element]
- 3 Select [ ] Select All
- 4 Rotation Axis : [Z Axis]
- 5 Select [Angle/Times]
- 6 Angle : "10", Number of Times : "36"
- 7 Mesh Set : [Silo]
- 8 Click [OK] Button









- Click Right Mouse Button and Select [Hide All]
- 2 Click [Front-Back Color]
- 3 Mesh > Element > **Parameters**
- 4 Select [2D] tab
- 5 Select [Change Csys]
- 6 Select [ 🔊 ] Select All
- Select [Reverse Normal] Option
- 8 Click [OK] Button



![](_page_5_Picture_11.jpeg)

2

![](_page_5_Picture_12.jpeg)

![](_page_5_Picture_13.jpeg)

![](_page_6_Picture_0.jpeg)

- 1 Click [Create Isotropic] Button
- 2 ID : "1" , Name : "Mat"
- 3 Elastic Modulus : "2e11" N/m<sup>2</sup>
- 4 Poisson's Ratio : "0.32"
- 5 Weight Density : "802\*9.80665" N/m<sup>3</sup>
- 6 Click **[OK]** Button
- 7 Click [Close] Button

![](_page_6_Picture_9.jpeg)

ID 1 Name Mat	2	
Model Type Elastic	······································	
Seneral Thermal Time Dependent To	emperature Deper	ndent
		\
Elastic Modulus(E)	2e+011	N/m²
Inc. of Elastic Modulus	0	N/m³
Inc. of Elastic Modulus Ref. Height	0	m
Poisson's Ratio(v)	0.32	4
Unit Weight(y) 5	7864.9333	N/m³
Basic Properties	tInput OU	sing Code
- Initial Stress Parameters		
Ko Determination	1	
Automatic		
Manual	Anisotropy	HE
Thermal Parameter		
Thermal Coefficient	1e-006	1/[T]
Molecular vapor diffusion coefficient	0	m²/sec
Thermal diffusion enhancement		in yood
mermar dir dsion enhancement	0	
Damping Ratio(For Dynamic)		
Damping Ratio	0.05	
Safety Result(Mohr-Coulomb)		
Cohesion(C)	30000	N/m²
Frictional Angle(Φ)	36	[deg]
Tensile Strength	0	N/m²

![](_page_7_Picture_0.jpeg)

![](_page_7_Figure_1.jpeg)

![](_page_8_Picture_0.jpeg)

# Static Analysis > Boundary > **Constraint**

- Select [36 Bottom] Nodes 1 (See Figure)
- 2 Click all Buttons
- Boundary Set : [Support] 3
- Click [OK] Button 4

Constraint			×
Basic Adv	anced Auto		
Name	Constraint-1		
Object			
Туре	Node		$\sim$
	Selected	36 Object(s) 1	
	OF	Symmetric Pla	ane
2 ☑ Tx ☑	]Ty ⊡Tz	XY Y	z
⊠Rx ⊻	Ry <b>⊘</b> Rz	ZX	tric
Boundary Set	Support		~ 🥸
k 🖉 🖉 4	ОК	Cancel	Apply

![](_page_8_Figure_8.jpeg)

![](_page_8_Picture_9.jpeg)

![](_page_9_Picture_0.jpeg)

![](_page_9_Figure_2.jpeg)

![](_page_9_Figure_3.jpeg)

 $\times$ 

3

4 5

Independent Var. Z

20 Inc.

10 12 14 16 18

OK

Closest Value

Cancel

Apply

2

6

10+Z\*Z Calculate

![](_page_10_Picture_0.jpeg)

## Static Analysis > Static Load > **Press.**

## **Procedure**

- Click [Top] view 1
- Click [Face] tab 2
- Object Type : [2D Element] 3
- Select [738 Right Elements] 4
- Direction Type : [Ref. CSys] 5
- Ref. CSys : [Global Rectangular X] 6
- Check off 7

### [Uniformly Distributed Load]

- Base Function : [Pressure] 8
- P1~P4 : "-1" 9
- Load Set : [Load] 10
- 1 Click [OK] Button

### Pressure × 2 Face Axisymmetric Name Pressure-1 Objec 3 2D Element Type Selected 738 Object(s) 4 Direction Ref. CSys ype 5 Ref. CSys Global Rectangular 🗸 🗸 ÷Č ۰x ΟY Οz Magnitude Uniformly Distributed Load 8 Base Function Pressure P or P1 -1 N/m<sup>2</sup> 9 -1 N/m<sup>2</sup> P2 -1 N/m² P3 P4 -1 N/m<sup>2</sup> ~ 🕹 Load Set Load 👳 🥒 (11 OK Cancel Apply

![](_page_10_Figure_16.jpeg)

![](_page_11_Picture_0.jpeg)

2

3

4

5

6

#### Procedure Add/Modify Analysis Case × Analysis Case Setting Title Stress Evaluation Title : "Stress Evaluation" Description ÞÐ Analysis Control . . . . . . . . . . . . . . . 2 Solution Type ÞÐ Linear Statio Output Control Solution Type : [Linear Static] ..... Construction Stage Set Drag & Drop [Static Load] and Analysis Case Model << >> All Sets Active Sets 🎒 Mesh 🖃 🎒 Mesh [Boundary Condition] to [Active Default Mesh Set Default Mesh Set 3 🛓 😻 Boundary Condition i 🖨 🕪 🖪 Support 🕹 Static Load 鬡 Support Sets] Window \$ tatic Loa 😻 Load ille 🥹 🕹 -🔀 Contact Pair 🛛 🛛 Contact Pair Click [OK] Button Analysis > **Perform** Click [OK] Button OK (4) Solve Each Load Set Independently Cancel Apply Sorting FEA NX Solver

FEA NX Solver	r			×
	Name	Туре	Description	
<b>V</b>	Stress Evaluation	Linear Static		
Check On/	Off	6	OK Cancel	

Results : Stress Evaluation > Linear Static > **Shell Element Stresses** 

### **Procedure**

Step

- 1 Double Click [S-MAX SHEAR TOP]
- Select [Deformed] for Mesh Shape at [Result] Tab

![](_page_12_Picture_4.jpeg)

![](_page_12_Figure_5.jpeg)

![](_page_12_Figure_6.jpeg)

![](_page_12_Picture_7.jpeg)