# Solid Mechanics (plate/shell, 3D)

Computational Design Laboratory Department of Automotive Engineering Hanyang University, Seoul, Korea



Copyright © 2019 Computational Design Lab. All rights reserved.

#### CONTENTS

- Plate and Shell Model
  - Thick plate
  - Hood
- 3D Model

#### **SHELLS AND PLATES**

#### Shells and Plates

The Shell interface ( ) is intended for the structural analysis of thin-walled structures. The formulation used in the Shell interface is a Mindlin-Reissner type, which means that transverse shear deformations are accounted for, and it can therefore be used for rather thick shells. It is also possible to prescribe an offset in the direction normal to a selected surface. The Shell interface also includes other features such as damping, thermal expansion, and initial stresses and strains. The preset studies available are the same as for the Solid Mechanics interface.

The Plate interface ( ) is the 2D analogy to the 3D Shell interface. Plates are similar to shells but act in a single plane and usually only with out-of-plane loads.

#### NAFEMS BENCHMARK: LE10

Thick Plate – Pressure

- Analysis Type: Linear elastic solid vs. plate
- Geometry: meters, thickness 0.6



COMSOL Structural Mechanics Verification Examples: Thick Plate Stress Analysis

CAE

- Loading
  - Uniform normal pressure of 1 MPa on the upper surface of the plate
- Boundary conditions
  - Face DCD'C': zero y-displacement
  - Face ABA'B': zero x-displacement
  - Face BCB'C': x and y-displacement fixed, z-displacements fixed along mid-plane
- Material properties
  - Isotropic: E = 210 GPa, v = 0.3
- Element types
  - Solid hexahedra, wedges and tetrahedra



• Meshes



- Output
  - Direct stress ( $\sigma_{yy}$ ) at point D: -5.38 MPa

# **LE10 COMPARISON**

1 = 10		integration		coarse	mesh	fine m	iesh
	order	points	element	s22	error	s22	еггог
Abaqus *	2	8	C3D20R	-7.93E+06	47.40%	-5.53E+06	2.79%
Ansys	2	8	SOLID95	-5.36E+06	-0.32%	-5.61E+06	4.26%
CalculiX	2	8	C3D20R	-5.36E+06	-0.32%	-5.61E+06	4.26%
Tahoe	2	8	hexahedron				
Abaqus *	2	27	C3D20	-6.72E+06	24.91%	-5.64E+06	4.83%
Ansys	2	14	SOLID95	-5.40E+06	0.46%	-5.61E+06	4.33%
CalculiX	2	27	C3D20	-5.20E+06	-3.32%	-5.50E+06	2.18%
Tahoe	2	27	hexahedron				

Structural Mechanics

Study : Stationary

# **ENVIRONMENT SETTING**

Select Study Stationary	2D
Image: Prestressed Analysis, Eigentrequency       Stationary         Image: Stationary       Prestressed Analysis, Frequency Domain         Image: Stationary       Prestressed Analysis, Frequency Domain         Image: Stationary       Prestressed Analysis, Eigentrequency         Image: Stationary       Image: Stationary         Image: Stationary       Image: Stat	ech ona
Added study: Stationary Added physics interfaces: Plate (plate)	
Part Help 🔀 Cancel 💟 Done	
780 MB   959 MB	

# **GEOMETRY CREATION**







Difference 기능을 이용해 구멍 생성



#### Rectangle 1 생성

Width: 3.25 Height: 2.75



Intersection 기능을 사용하여 다음과 같은 형상 생성

#### THICKNESS



#### MATERIAL PROPERTY



# **BOUNDARY CONDITION**



# **BOUNDARY CONDITION**



#### LOADING CONDITION



#### MESH



Mapped mesh의 Distribution 기능을 이용하 여 요소 생성

Distribution 1  $\rightarrow$  3,4 bnd. elements: 3 Distribution 2  $\rightarrow$  1,2 bnd. elements: 2

Compute를 클릭하여 해석 수행

#### **POST-PROCESSING**



Derived Values 의 Point Evaluation 기능을 이용하여 3번 절점의 plate.Syy 응력 을 계산

plate.SYY: Second Piola-Kirchhoff stress, yy component

결과값: -4.61 MPa

**※ Ref : -5.38 MPa** 

#### MESH



Fine mesh로 변경 Distribution 1 → 3,4 bnd. elements: 6 Distribution 2 → 1,2 bnd. elements: 4 Compute를 클릭하여 해석 수행 결과값: -4.98 MPa **※ Ref : -5.38 MPa** 

#### SUMMARY

1 5 10		integration		coarse	mesh	fine m	iesh
	order	points	element	s22	error	s22	error
Abaqus *	2	8	C3D20R	-7.93E+06	47.40%	-5.53E+06	2.79%
Ansys	2	8	SOLID95	-5.36E+06	-0.32%	-5.61E+06	4.26%
CalculiX	2	8	C3D20R	-5.36E+06	-0.32%	-5.61E+06	4.26%
Tahoe	2	8	hexahedron				
Abaqus *	2	27	C3D20	-6.72E+06	24.91%	-5.64E+06	4.83%
Ansys	2	14	SOLID95	-5.40E+06	0.46%	-5.61E+06	4.33%
CalculiX	2	27	C3D20	-5.20E+06	-3.32%	-5.50E+06	2.18%
Tahoe	2	27	hexahedron				

COMSOL	2	plate	-4.61E+06	14.3 %	-4.98E+06	7.4 %
COMSOL	2	hexahedron				

# **ENVIRONMENT SETTING**

🖻   🗅 📂 🗔 🔊 ち ぐ 咱 后 唔 前 訳 羽 民 -	Untitled.mph - COMSOL Multiphysics	
File	lesh Study Results Developer	
Select Study	Stationary study is used when field variables do not change over time. Examples: In electromagnetics, it is used to compute static electric or magnetic fields, as well as direct currents. In heat transfer, it is used to compute the temperature field at thermal equilibrium. In solid mechanics, it is used to compute deformations, stresses, and strains at static equilibrium. In fluid flow it is used to compute the steady flow and pressure fields. In chemical species transport, it is used to compute steady-state chemical composition in steady flows. In chemical reactions, it is used to compute the chemical composition at equilibrium of a reacting system. It is also possible to compute several solutions, such as a number of load cases, or to track the nonlinear response to a slowly varying load.	
Added study: Stationary Added physics interfaces: Solid Mechanics (solid)		
Physics		
Help 🛛 Cancel 💟 Done		
1	.18 GB   1.34 GB	

Dimension : 3D
 Physics :
 Structural Mechanics
 → Solid Mechanics
 Study : Stationary

Done 클릭

🍳   🗅 📂 🔛 🐼 🕨 🐄 🖒 🛤		Untitled.mph - COMSOL Multiphysics	
File - Home Definitions Geome	etry Materials Physics Mesh Study Resi	Ilts Developer	?
Application Builder Application Application Model Pi Parameter a= Variables ftx Functions Definitions	rs Build All Geometry	dd ysics         Mesh Wesh         1 - Mesh         Compute Study         Add Study         Select Plot         Add Plot           Group - Mesh         Study         Study         Results	Windows Reset • Desktop • Layout
Model Builder       Image: State of the st	Settings • • • Work Plane Build Selected • • Build All Objects Label: Work Plane 1 • Plane Definition Plane type: Quick • • Plane: xy-plane • Offset type: Distance • z-coordinate: 0 m b Local Coordinate System • Unite Objects Repair tolerance: Automatic • • Selections of Resulting Entities Contribute to: None • New Resulting objects selection Show in physics: Boundary selection - Selections from plane geometry Show in physics	Graphics Q Q : 1 → 1 → 1 → 1 → 1 → 1 → 1 → 1 → 1 → 1	- + + × - + ×
	1.13 GB   1.28 GB		

Geometry 1 마우스 우클릭 → Work Plane 클릭





Extrude 메뉴를 이용하여 Z 방향으로 0.6 m 두께를 가지는 형상 생성

# MATERIAL PROPERTY



Linear Elastic Material 클릭

2 E: 210e9 mu: 0.3 rho : 0 입력

# **BOUNDARY CONDITION**



#### Symmetry 경계조건 생성

1, 6번 경계 선택

# **BOUNDARY CONDITION**



#### Fixed 경계조건 생성 (왼쪽버튼 누른 채로 회전)

2 5번 경계 선택

### LOADING CONDITION



#### MESH



Mapped mesh의 Distribution 기능을 이용 4번 평면에 요소 생성

#### MESH



Swept 메뉴를 이용하여 전체 영역에 한 개의 layer를 가지는 요소 생성

Compute를 클릭하여 해석 수행

### **POST-PROCESSING**



Derived Values 의 Point Evaluation 기능을 이용하여 6번 절점의 solid.sy 응력을 계산

결과값: -4.10 MPa

※ Ref: -5.38 MPa

#### **MESH**



Fine mesh로 변경 Compute를 클릭하여 해석 수행

결과값: -5.13 MPa

※ Ref: -5.38 MPa

#### SUMMARY

1 = 10		integration		coarse	mesh	fine m	esh
	order	points	element	s22	еггог	s22	error
Abaqus *	2	8	C3D20R	-7.93E+06	47.40%	-5.53E+06	2.79%
Ansys	2	8	SOLID95	-5.36E+06	-0.32%	-5.61E+06	4.26%
CalculiX	2	8	C3D20R	-5.36E+06	-0.32%	-5.61E+06	4.26%
Tahoe	2	8	hexahedron				
Abaqus *	2	27	C3D20	-6.72E+06	24.91%	-5.64E+06	4.83%
Ansys	2	14	SOLID95	-5.40E+06	0.46%	-5.61E+06	4.33%
CalculiX	2	27	C3D20	-5.20E+06	-3.32%	-5.50E+06	2.18%
Tahoe	2	27	hexahedron				

COMSOL	2	plate	-4.61E+06	14.3 %	-4.98E+06	7.4 %
COMSOL	2	hexahedron	-4.10E+06	23.8 %	-5.13E+06	4.6 %

#### **HOOD PANEL ANALYSIS**

Panel stiffness: load(1N)/deflection

$$K = \frac{CEt^2}{R\sqrt{1-v^2}} \text{ where curvature } \frac{1}{R} = \frac{\left(\frac{L_1^2}{R_1}\right) + \left(\frac{L_2^2}{R_2}\right)}{2L_1L_2}$$

$$\begin{cases} C: \text{ constant (2.309)} \\ t: \text{ panel thickness} \\ R: \text{ spherical radius} \end{cases}$$

- $L_1, L_2$ : rectangluar panel dimensions
- $R_1, R_2$ : panel radii of curvature in orthogonal directions
- Oil-can load: buckling load

$$P_{cr} = \frac{CR_{cr}\pi^{2}Et^{4}}{L_{1}L_{2}\left(1-\nu^{2}\right)} \text{ where } \begin{cases} R_{cr} = 45.929 - 34.183\lambda + 6.397\lambda^{2} \\ \lambda = 0.5\sqrt{\frac{L_{1}L_{2}}{t}\sqrt{\frac{12\left(1-\nu^{2}\right)}{R_{1}R_{2}}}} \\ C = 0.645 - 0.775L_{1}L_{2} \end{cases}$$

valid over the range  $\frac{R_1}{L_1}$  and  $\frac{R_2}{L_2} > 2$ ,  $\frac{1}{3} < \frac{L_2}{L_1} < 3$ ,  $L_1 L_2 < 0.774 m^2$ 



K = 28.81 N/mm $P_{cr} = 50.53 \text{ N}$ 

t=

# **ENVIRONMENT SETTING**

0   <u>} </u>	Untitled.mph - COMSOL Multiphysics	V F
File       Home       Definitions       Geometry       Materials       Physics       I         Select Studies         Image: Stationary       <	Until ed.mph - COMSOL Multiphysics           Weth         Study         Results         Developer        Stationary study is used when field variables do not change over time.       Examples: In electromagnetics, it is used to compute static electric or magnetic fields, as well as direct currents. In heat transfer, it is used to magnetic fields, as well as direct currents. In heat transfer, it is used to mechanics, it is used to compute the transparture field at thermal equilibrium. In solid mechanics, it is used to compute the transpart is used to compute the transpart is used to compute the transpart is used to compute steady flows. In chemical reactions, it is used to compute the chemical composition in steady flows. In chemical reactions, it is used to compute the chemical composition at equilibrium of a neacting system.       It is also possible to compute several solutions, such as a number of load cases, or to track the nonlinear response to a slowly varying load.	
Physics       Physics       Physics       Help     X Cancel		
	1.21 GB   1.33 GB	

Dimension : 3D
 Physics :
 Structural Mechanics
 → Shell
 Study : Stationary

Done 클릭

# **GEOMETRY IMPORT**



Geometry에서 단위를 mm 로 설정

2 Mesh 우클릭 → Import

Settings 화면에서

1) Source : NASTRAN file

2) Browse 버튼 클릭해서 hood\_center.nas 파일 선택

3) Boundary partitioning: Minimal

4) Import 클릭

# MATERIAL PROPERTY



# MATERIAL PROPERTY



# **BOUNDARY CONDITION**



#### Pinned경계조건 생성

2 최외각 경계 선택

# LOADING CONDITION



#### **MESH**



Study → Compute 클릭 해석 수행

#### **POST-PROCESSING**



### LINEAR BUCKLING



Toolbar에서 Study → Add Study 클릭

<sup>2</sup> Linear Buckling 추가

Compute를 클릭하여 해석 수행

#### **POST-PROCESSING**



Critical load factor : 53.015

결과값: 53.015 N

#### Analytic solution

$$P_{cr} = \frac{CR_{cr}\pi^2 Et^4}{L_1 L_2 \left(1 - \mu^2\right)} = 50.53 \text{ N}$$

#### NAFEMS BENCHMARK: LE11

Solid Cylinder/Taper/Sphere – Temperature

- Analysis Type: Linear elastic solid
- Geometry: meters



CAE

- Loading
  - Linear temperature gradient in the radial and axial direction
  - $T (^{\circ}C) = (x^2 + y^2)^{1/2} + z$
- Boundary conditions
  - Symmetry on xz-plane
  - Symmetry on yz-plane
  - Face on xy-plane: zero z-displacement
  - Face HIH'I': zero z-displacement
- Material properties
  - Isotropic: E = 210 GPa, v = 0.3
  - $\alpha = 2.3 * 10^{-4} / C$
- Element types
  - Solid hexahedra, wedges and tetrahedra

• Meshes



• Output

– Direct stress ( $\sigma_{zz}$ ) at point A: -105 MPa

# **ENVIRONMENT SETTING**

File  Home Definitions Geometry Materials Physics Mes	n Study Results Developer	
Select Study	Stationary study is used when field variables do not change over time. Examples: In electromagnetics, it is used to compute static electric or magnetic fields, as well as direct currents. In heat transfer, it is used to compute the temperature field at thermal equilibrium. In solid mechanics, it is used to compute deformations, stresses, and strains at static equilibrium. In fluid flow it is used to compute the steady flow and pressure fields. In chemical species transport, it is used to compute the chemical composition in steady flows. In chemical reactions, it used to compute the chemical composition at equilibrium of a reacting system. It is also possible to compute several solutions, such as a number of load cases, or to track the nonlinear response to a slowly varying load.	
Physics Physics Cancel Conce		
1.18	GB   1.34 GB	

Dimension : 3D Physics : Structural Mechanics → Solid Mechanics Study : Stationary

Done 클릭

# **GEOMETRY CREATION**

Image: Source of the series in type	Work Plane에서 Plane을 xz-plane으로 변경
Step 1: Stationary   Solver Configurations   Job Configurations   Selections of Resulting Entities   Contribute to:   Nome   Resulting objects selection   Show in physics:   Boundary selection   Show in physics:   Show in physics     Messages   Progress   Image: Selection from plane geometry     Show in physics     Messages   Progress   Image: Selection from plane geometry     Show in physics     Messages   Progress   Image: Selection from plane geometry     Show in physics     Messages   Progress   Image: Selection from plane geometry     Show in physics     Messages   Progress   Image: Selection from plane geometry     Show in physics     Messages   Progress   Image: Selection from plane geometry     Show in physics     Messages   Messages   Image: Selection from plane geometry     Show in physics     Messages     Messages     Messages     Messages     Messages     Messages     Messages     Messages     Messages     Messages <td></td>	



Plane Geometry 우클릭 → Import

Settings 화면에서

1) Source : DXF file

2) Browse 버튼 클릭해서 LE11.dxf 파일 선택

3) Import 클릭



Revolve 메뉴를 이용하여 90도 회전시킨 형상을 생성

# MATERIAL PROPERTY



#### Linear Elastic Material 클릭

2 E: 210e9 mu: 0.3 rho : 0 입력

# MATERIAL PROPERTY



# **BOUNDARY CONDITION**



# **BOUNDARY CONDITION**



Prescribed Displacement 경계조건 생성

2 4(위쪽), 5(아래쪽)번 경계 선택 후 z방향 변위에 0입력

#### MESH



Mapped mesh 와 Swept 기능을 이용하여 요소 생성

Compute를 클릭하여 해석 수행

## **POST-PROCESSING**



Derived Values 의 Point Evaluation 기능을 이용하여 10번 절점의 solid.sz 응력을 계산

결과값: -97.1 MPa

**※ Ref : -105 MPa** 

#### **MESH**



Fine mesh로 변경 후 해석 실행

결과값: -103.1 MPa

**※ Ref : -105 MPa** 

# **ENVIRONMENT SETTING**

Norm         Definition         Generity         Material         Physics         Meth         Study         Results         Developer           Select Studies <b></b>	🗅 📂 🖬 💫 ト ゥ ぐ 咱 市 由 前 尿 初 民 - I	Untitled.mph - COMSOL Multiphysics	
Select Study ■ of preset Studie ■ disperingency Trepuncy-Domain Medal Bigeringency Trepuncy-Domain Medal ■ disperingency ■ disperingen	e Home Definitions Geometry Materials Physics	Mesh Study Results Developer	?
Y Help Cancel Done	Select Study   Preset Studies  Both Pre-Tension  Frequency Domain  Frequency Domain	<text><text><text><text></text></text></text></text>	

Dimension : 2D Axissym.
 Physics :
 Structural Mechanics
 → Solid Mechanics
 Study : Stationary

Done 클릭



# MATERIAL PROPERTY



# MATERIAL PROPERTY



# **BOUNDARY CONDITION**



Prescribed Displacement 경계조건 생성

2, 3번 경계 선택 후 z 방향 변위에 0 입력

#### MESH



Mapped mesh 기능을 이용 하여 요소 생성

Compute를 클릭하여 해석 수행

#### **POST-PROCESSING**



Derived Values 의 Point Evaluation 기능을 이용하여 3번 절점의 solid.sz 응력을 계산

결과값: -97.3 MPa

**※ Ref : -105 MPa** 

#### MESH



Fine mesh로 변경 후 해석 실행

결과값: -103.7 MPa

**※ Ref : -105 MPa** 

#### SUMMARY

• Meshes



Software	Order	Element	Coarse mesh		Fine mesh	
			Result	Error	Result	Error
COMSOL	2	axisymmetry	-97.3	7.33 %	-103.7	1.24 %
COMSOL	2	hexahedron	-97.1	7.52 %	-103.1	1.81 %

# **HOMEWORK: SOLID MECHANICS**

#### Tapered Thick Shell under Pressure and Gravity Loads

Determine the hoop stress at point A of a tapered thick shell subjected to the following loading conditions: Study A. Uniform normal pressure of 100 MPa on outer face highlighted in green. Study B. Uniform acceleration - 9.81 m/s<sup>2</sup> in global y-direction. The bottom face on the xz plane is constrained in the vertical y-direction.



Modulus of elasticity (E) = 210 GPa Poisson's ratio (v) = 0.3 Density( $\rho$ ): 7000 kg/m<sup>3</sup>

Von Mises stress at point A

Study A : 211.6 MPa , Study B : 0.031 MPa

Study case 1 : 3D Solid Model Study case 2 : 2D Axisymmetric Model