

Explicit Dynamic Analysis (2)

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



한양대학교
HANYANG UNIVERSITY

CDL Computational
Design
Lab

OUTLINE

- **Lecture Goals**

- ✓ 차량 프레임 부재(frame member)의 외연적 동해석(explicit dynamic analysis)을 수행하고 충돌안전(crashworthiness) 분석을 위한 FOA(First Order Analysis) 모델과 FE(Finite Element) 모델의 붕괴 거동(collapse behavior)을 확인한다.

- **Contents**

- ✓ Joint moment analysis
- ✓ Mid rail analysis
 - : beam element
 - : shell element

- 해석 프로세스

- 기하형상 생성
- 재료 물성 및 특성 입력
- 요소망 생성
- 구속조건 설정
- 하중조건 설정
- 해석케이스 정의 및 해석 실행
- 후처리

FIRST ORDER ANALYSIS: CRASHWORTHINESS

2004-01-1660

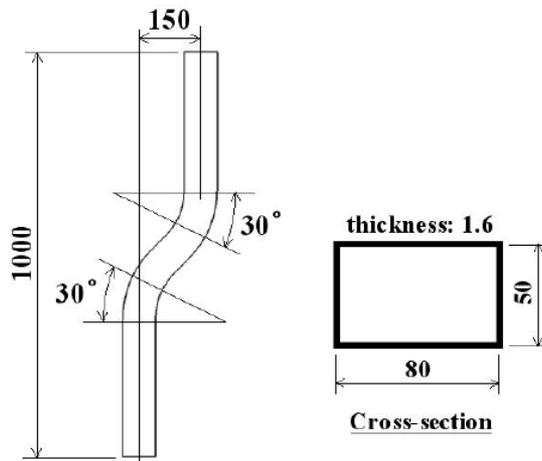
First Order Analysis for Automotive Body Structure Design - Part 3: Crashworthiness Analysis Using Beam Elements

Hidekazu Nishigaki
Toyota Central R&D Labs., Inc.

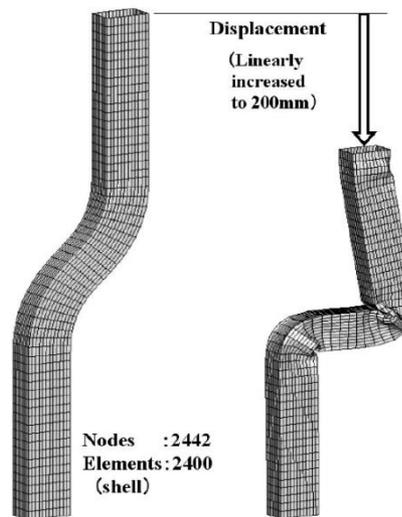
Noboru Kikuchi
The University of Michigan

Copyright © 2003 SAE International

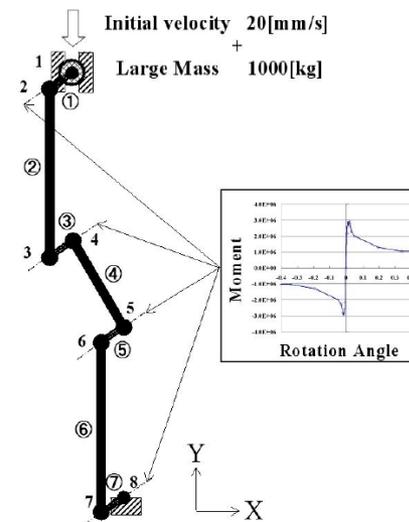
기하 형상 정보



셸 모델 해석



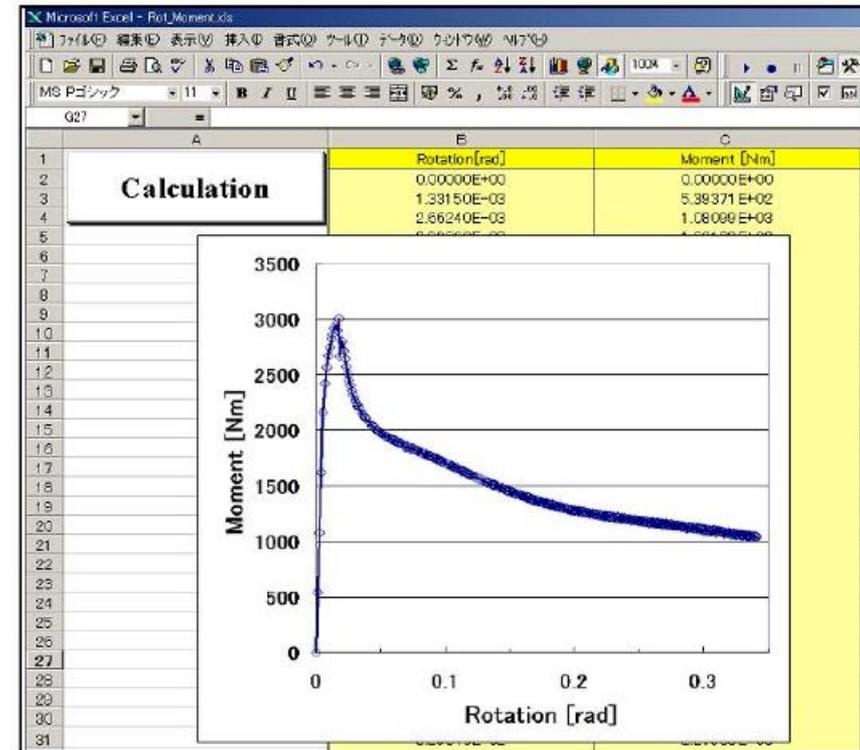
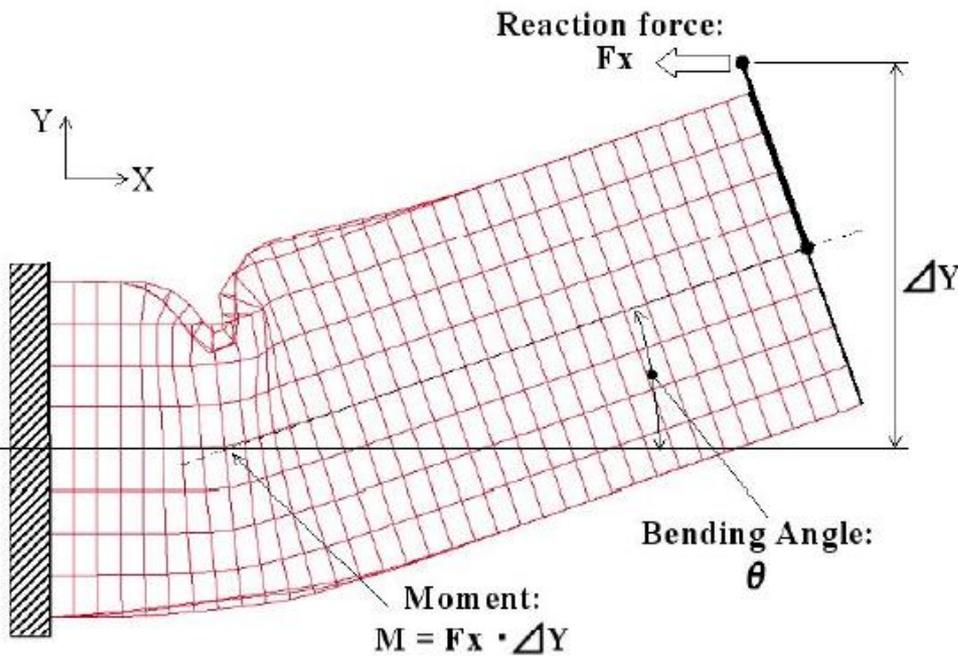
빔 모델 해석



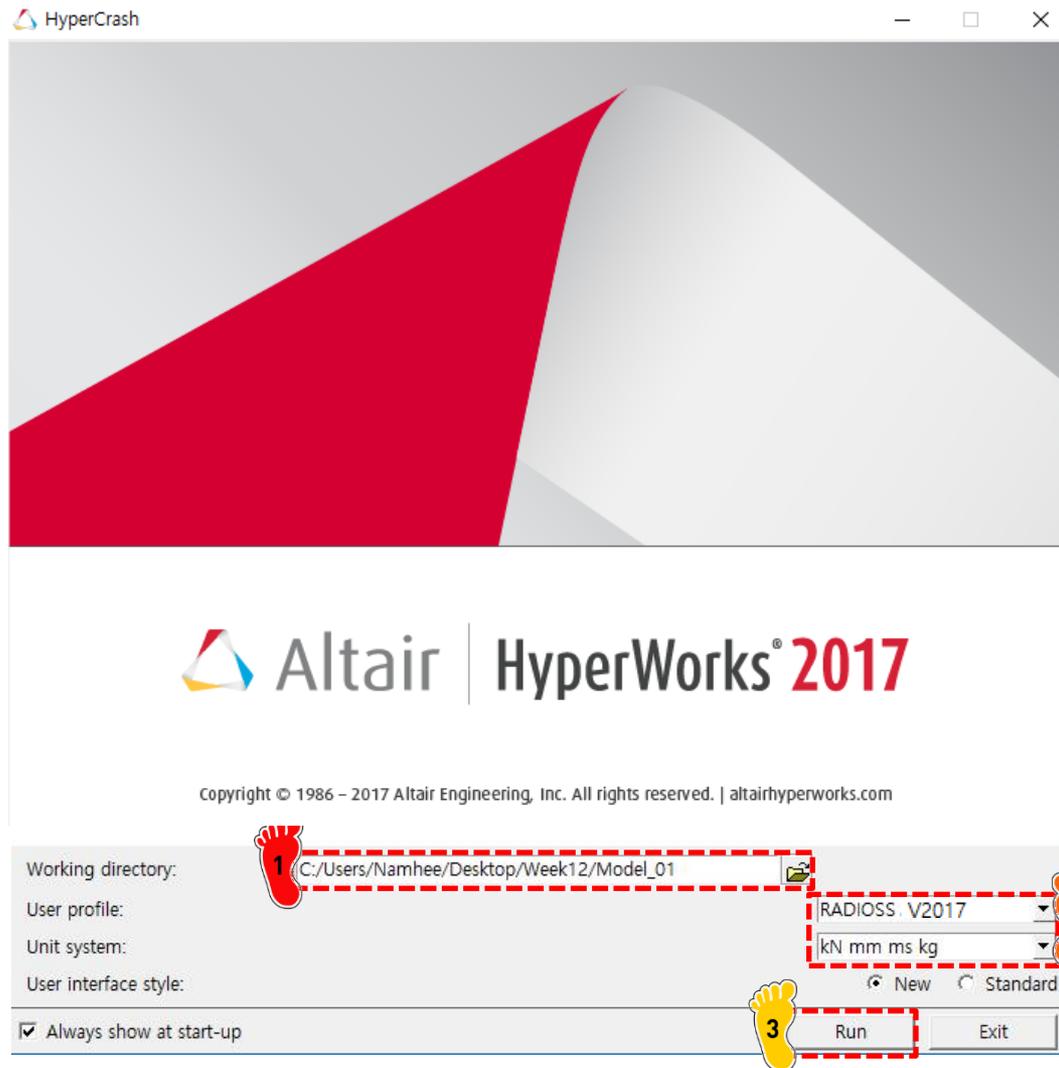
JOINT MOMENT ANALYSIS

예제: JOINT MOMENT ANALYSIS

- Shell 모델의 등가 강성 계산



HYPERCRASH 실행

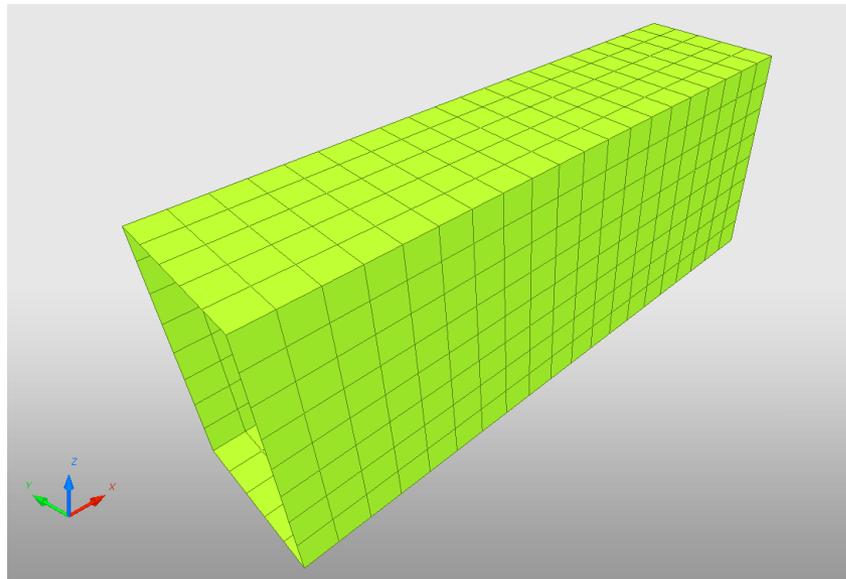
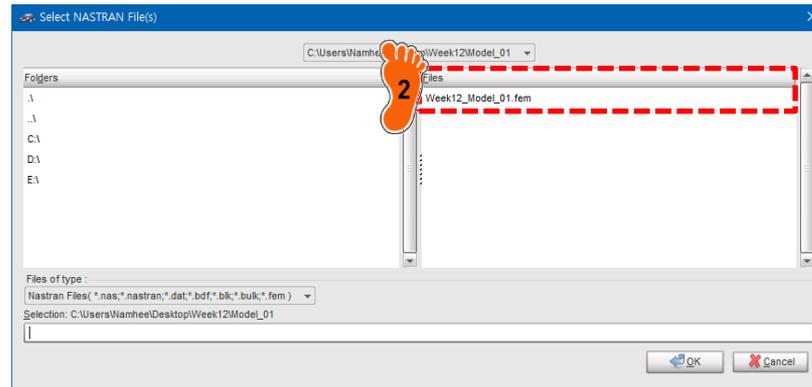
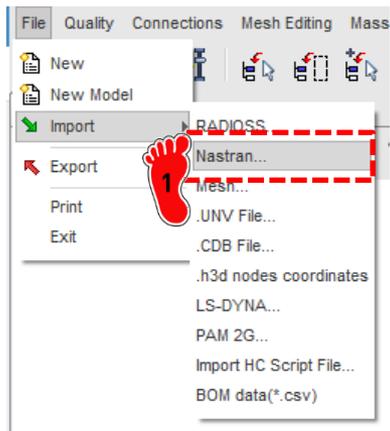


1 HyperCrash2017 실행 후
Working directory 설정
(모델 파일이 있는 폴더로
지정)

2 User profile, Unit system
설정

3 Run 클릭

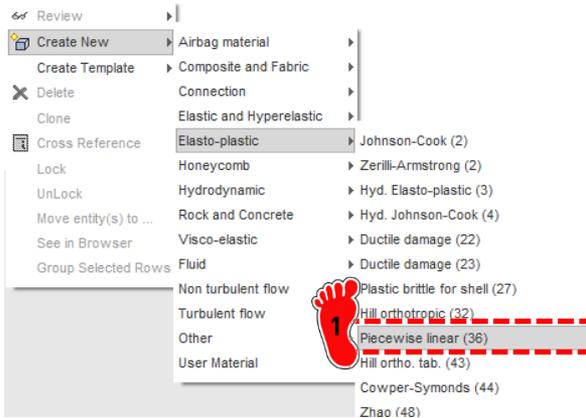
유한요소해석 모델 불러오기



1 File > Import > Nastran
클릭

2 모델 파일 클릭 후 불러오기
(Week12_Model_01.fem)

재료 물성 입력 (1)



an isotropic elasto-plastic material using user-defined functions (e.g. stress-strain curve)

PLAS_TAB	
ID	2
Title	New MAT 2
Local Unit System	None
[RHO_I] Initial density*	7.85E-6
[RHO_0] Reference density	0
[E] Young's modulus*	206
[Nu] Poisson's ratio*	.3
[Eps_p_max] Failure plastic strain	0
[Eps_t] Tensile failure strain	0
[Eps_m] Tensile failure strain	0
[C_hard] Hardening coefficient	0
[F_smooth] Strain rate smoothing	<input type="checkbox"/>
[F_cut] Cutoff frequency for strain rate filtering	0
[Eps_f] Maximum tensile failure strain	0
[VP] Strain rate choice flag	0: Strain rate are total strain rate
[fct_IDp] Pressure vs. yield factor function	None
[Fscale] Y-Scale factor for yield factor function	1
[Fct_IDE] Function identifier for the scale factor of You	None
[EInf] Saturated Young's modulus for infinitive plastic s	0
[CE] Parameter for Young's modulus evolution	0
Heat	<input type="checkbox"/>

1 Model > Material 클릭 > 마우스 우클릭 후 Create New > Piecewise linear(36) 재료 생성

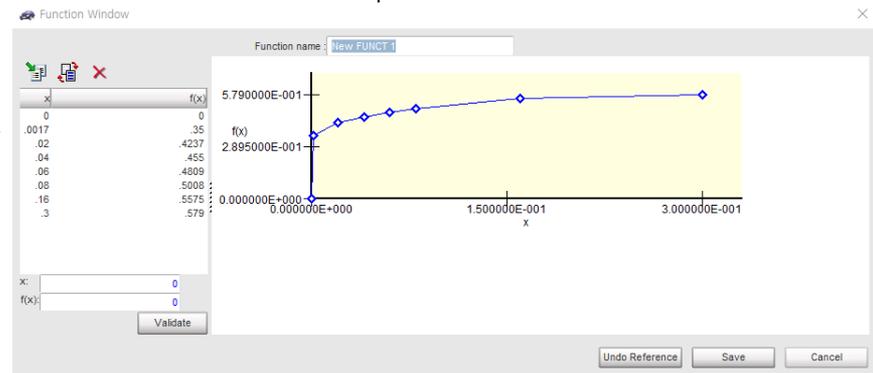
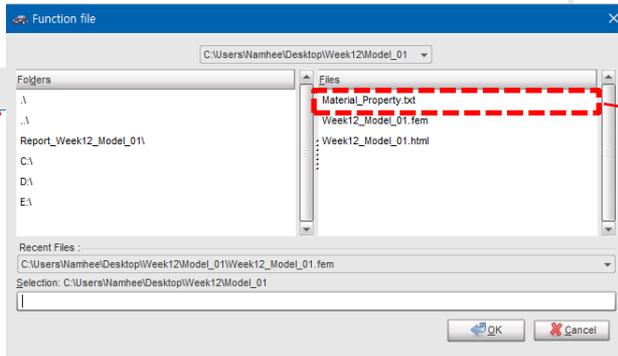
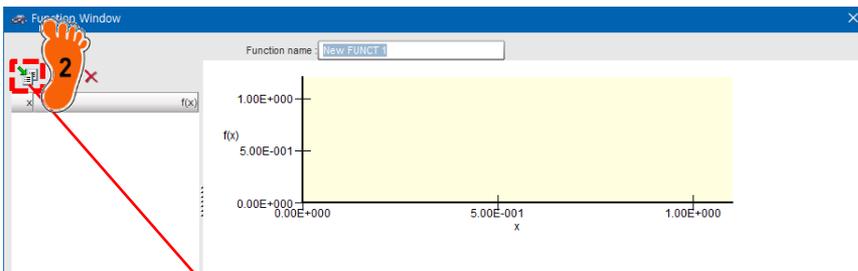
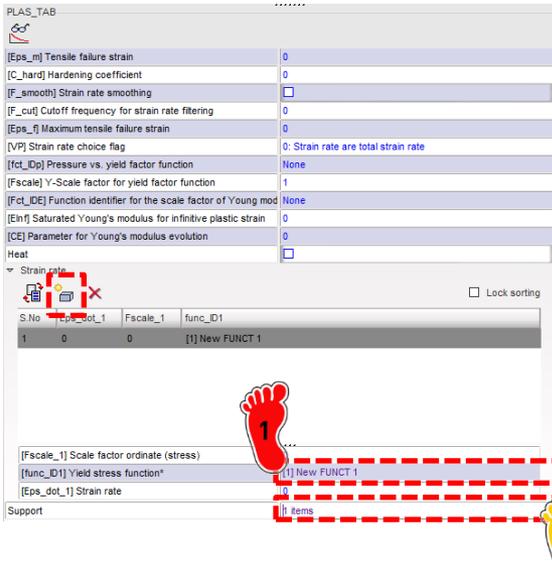
2 재료 물성치 입력
[RHO_I]: 7.85e-6
[E]: 240
[Nu]: 0.3

재료 물성 입력 (2)

1 Stress-Strain curve 입력
(마우스 우클릭 후 "Define function" 클릭)

2 "Import curve" 클릭 후
"Material Property.txt" 파일 불러오기

3 모델에 재료 적용



특성 입력

SHELL	
ID	2
Title	New PROP 2
Local Unit System	None
[lsmstr] Flag for shell small strain formulation	0: Use value in /DEF_SHELL
[lshell] Flag for 4 node shell element formulation	0: Use value in /DEF_SHELL
[lsh3n] Flag for 3 node shell element formulation	0: use value in /DEF_SHELL
[ldrill] Flag for drilling degree of freedom stiffness	0: No
[P_thick_fail] Percentage of through thickness integration	0
[hm] Shell membrane hourglass coefficient	0
[hf] Shell out of plane hourglass	0
[hr] Shell rotation hourglass coefficient	0
[dm] Shell membrane damping	0
[dn] Shell numerical damping	0
[N] Number of integration points through the thickness	0
[lstrain] Flag to compute strains for post-processing	0: Use value in /DEF_SHELL
[Thick] Shell thickness*	1.6
[Ashear] Shear factor	0
[thick] Flag for shell resultant stresses calculation	0: Default set to value defined with /DEF_SHELL
[lplas] Flag for shell plane stress plasticity	0: Default set to value defined with /DEF_SHELL
Support*	1 items

Close Save Cancel



Model > Property 클릭,
마우스 우클릭 후
Create New > Surface >
Shell (1) 클릭 >

두께 1.6mm 입력



모델에 특성 적용

Help:
/PROP/TYPE1 (SHELL)

RBE2 생성 (1)

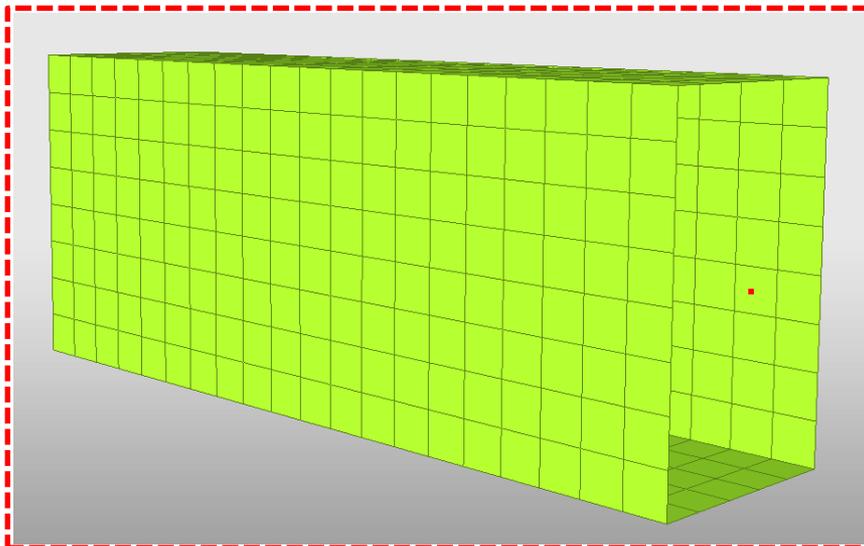
The image illustrates the first step of creating an RBE2 constraint. It shows the software interface with the 'Mesh Editing' menu open and 'RBE2' selected. The 'RBE2' dialog box is open, showing the 'Title' as 'New RBE2 1' and the '[Node_IDm] Master node identifier*' set to 'None'. A context menu is open over the 'None' field, with 'Create New' selected. Below this, the 'NODE' dialog box is shown with the ID set to 547 and coordinates (200, 25, 40). At the bottom, a 3D model of a meshed part is shown, with a red arrow pointing to the location where the master node is being defined.

1 Mesh Editing > RBE2 생성 > Create New

2 Master node 좌표:
(200, 25, 40)

3 Support: X방향 끝단 노드 선택

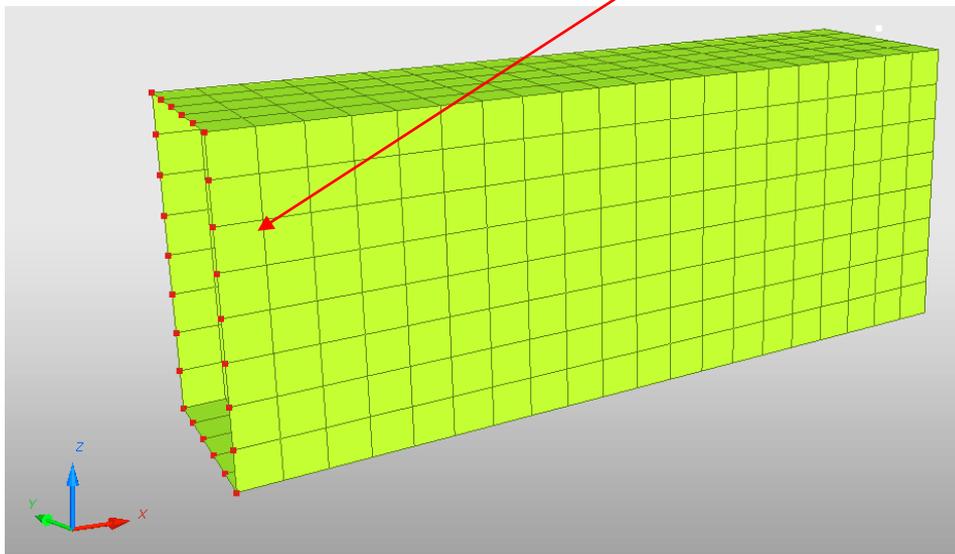
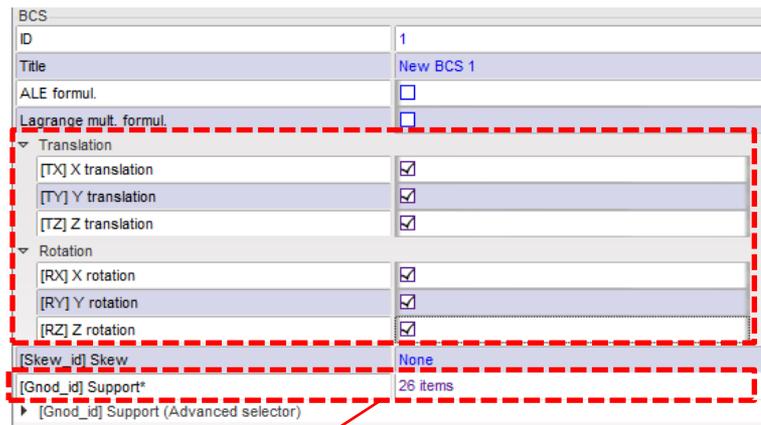
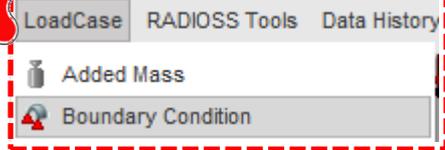
RBE2 생성 (2)



-  새로운 RBE2 생성
-  Master node 좌표:
(200, 25, 83.3)
-  Support: 이전에 만든 RBE2
mater node (no.547)

(F8 : free node display
on/off)

구속조건 설정



LoadCase > Boundary Condition 클릭, 마우스 우클릭 후 Create new 클릭

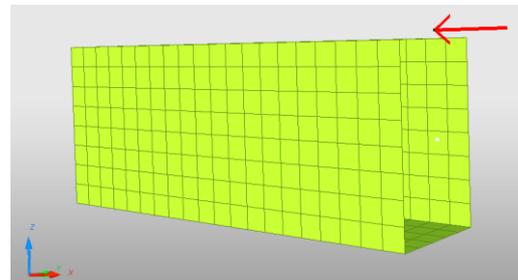
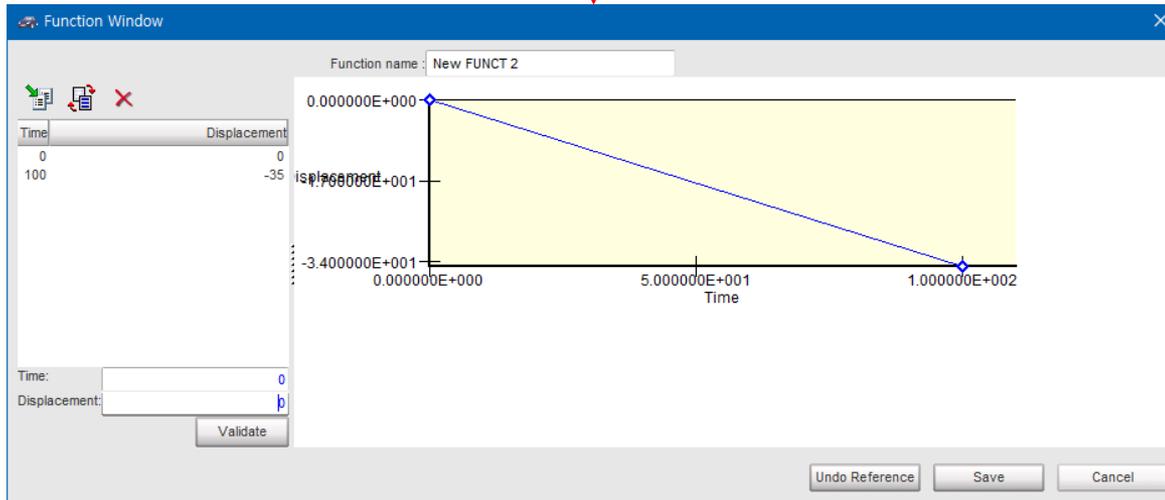
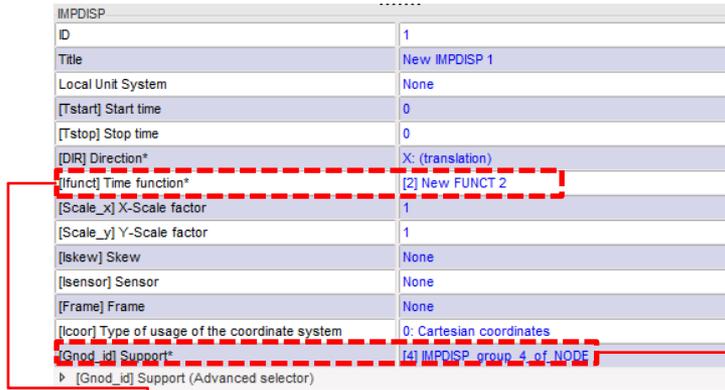
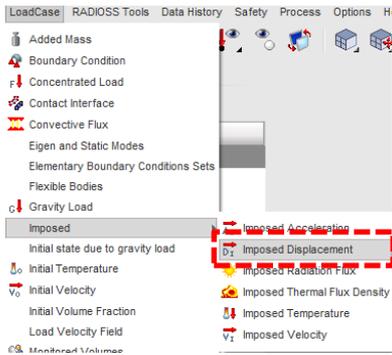


완전 구속 설정



구속할 절점 선택

변위조건 설정



1 LoadCase > Imposed > Imposed Displacement 클릭,
마우스 우클릭 후 Imposed displacements 생성

2 시간-변위 그래프 생성 (0,0), (100,-35)
DIR: X

3 Support: 앞서 만든 RBE2 요소의 master node 선택

접촉조건 설정

HyperCrash 접촉조건의 종류 (/INTER/TYPE#)

ALE/Lagrange with sliding (Type 1)

Kinematic condition (Type 2)

Surface/Surface (Type 3)

Nodes/Surface (Type 5)

Rigid body/Rigid body (Type 6)

Multi usage (Type 7)

Drawbeads (Type 8)

ALE/Lagrange with Void (Type 9)

Tied with void (Type 10)

Edge to edge (Type 11)

Fluid/Fluid (Type 12)

Ellips. surface/Node (Type 14)

Ellips. surface/Surface (Type 15)

Node/Solid (Type 16)

Solid/Solid (Type 17)

Fluid/Structure (Type 18)

Surface/Surface with edge treatment (Type 19)

Multi usage constant stiffness (Type 20)

Surface/Rigid surface (Type 21)

Airbag surface (Type 23)

Surface/Surface (Type 24)

Sub-interface

- Interface Type 7 is a general purpose interface and can simulate all types of impact between a set of nodes and a master surface.
- This interface can simulate self-impact, especially buckling during a high speed crash.
- Interface Type 7 solves all problems.
- The main advantage of interface Type 7 is that the stiffness is not constant and increases with penetration preventing the node from going through the shell mid-surface.

접촉조건 설정

LoadCase RADIOSS Tools Data History

- Added Mass
- Boundary Condition
- Concentrated Load
- Contact Interface**

Review

- Create New
 - ALE/Lagrange with sliding (Type 1)
 - Kinematic condition (Type 2)
 - Surface/Surface (Type 3)
 - Nodes/Surface (Type 5)
 - Rigid body/Rigid body (Type 6)
 - Multi usage (Type 7)**
- Create Template
- Delete
- Data History
- Clone
- Change Subtype

1 LoadCase > Contact Interface 클릭,
마우스 우클릭 후 "Multi usage (Type 7) 클릭

2

TYPE7
D	1
Title	New INTER 1
Self Impact	<input checked="" type="checkbox"/>
Local Unit System	None
Formulation	0: Classical
[lstf] Stiffness definition	0: Default, set to value defined in /DEFAULT/INTER/TYPE
[lthe] Heat contact	<input type="checkbox"/>
[lgap] Gap/element option	0: Default, set to value defined in /DEFAULT/INTER/TYPE
[Fpenmax] Maximum fraction of initial penetration	0
[lbag] Vent hole closure when contact	0: Default, set to value defined in /DEFAULT/INTER/TYPE
[ldel] Node and segment deletion	0: Default, set to value defined in /DEFAULT/INTER/TYPE
[lcurv] Slave gap with curvature	0: No curvature
[ladm] Local curvature flag	0: Not activated
[Stfac] Scale factor for stiffness	1
[dtmin] Limiting nodal time step	0
[lrem_gap] Flag for deactivating slave nodes if element s	0: Default, set to value defined in /DEFAULT/INTER/TYPE
[lrem_id] Flag for deactivating the slave node, if the sam	0: default, set to the value defined in /DEFAULT/INTER/T
[Gapmin] Min. gap for impact activ.	.9
[_BC] Bound. cond. deactivation	
[inact] Stiffness deactiv. (init. penetration)	0: Default, set to value defined in /DEFAULT/INTER/TYPE
[VisS] Critical damping coeff. on interface stiffness	0
[VisF] Critical damping coeff. on interface friction	0
[Tstart] Start time	0
[Tstop] Stop time	0
[Bumult] Sorting factor	0
[lfric] Friction formulation	0: Static Coulomb
[Fric] Coulomb friction	0.2
[lform] Friction penalty formulation	0: Default, set to value defined in /DEFAULT/INTER/TYPE
[lfiltr] Friction filtering	0: No filtering
[sens_ID] Sensor to Activate/Deactivate the interface	None
[fct_ID] Friction coefficient with temperature function id	None
[AscaleF] Abscissa scale factor on FCT_IDK	0
[fric_ID] Friction identifier for friction definition for select	None
[Mast_id] Master surface*	[8] INTER_group_8_of_PART
[Mast_id] Master surface (Advanced selector)	

2 Self Impact 체크
[Stfac] : 1
[Gapmin]: 0.9
[Fric]: 0.2
[Mast_id]: 모델 선택

Interface stiffness scale factor
(1.0 real: contact stiffness is taken from the master segment)

Minimum gap for activation of interface

해석 케이스 설정

1 Model LoadCase RADIOSS Tools Dat

Browser

Part

Material

Property

Add to Material

Friction

Function

Function 2D

Function Scale and Shift

Control Card

2 IOFLAG : Describes the input-output flags

IOFLAG	
[IPRI] Printout	0: Reduced
[IGTYP] Type of G00-file	0: Default
[IOUTP] Y000 file	0: Default
[IOUTYY] Ynnn file format	0: Default
[IROOTYY] Ynnn file name	0: Default
[IDROT] Force computation of rotational DOF flag	1: Force computation of rotational DOF

RUN_NAME_NUMBER_LETTER : Identifies the run number

RUN_NAME_NUMBER_LETTER	
Run Number	1
[DT_STOP] Final time for run	100
Restart letter	: Default

ANIM_DT : Write animation files

ANIM_DT	
[TSTART] Start time	0
[TFREQ] Time frequency	1

ANIM_VECT : Generates animation files containing vectorial data for the specified variable

ANIM_VECT_RESTYPE_SUPPORT	
Variable name to be saved in animation file*	FREAC: Reaction forces for imposed velocities, displac
Variable name to be saved in animation file*	DROT: Rotations

1 Model > Control Card 클릭

2 파라미터 설정

3 모델 export (RADIOSS 선택)

3

File Quality Connections Mesh Editin

New

New Model

Import

Export

Print

Exit

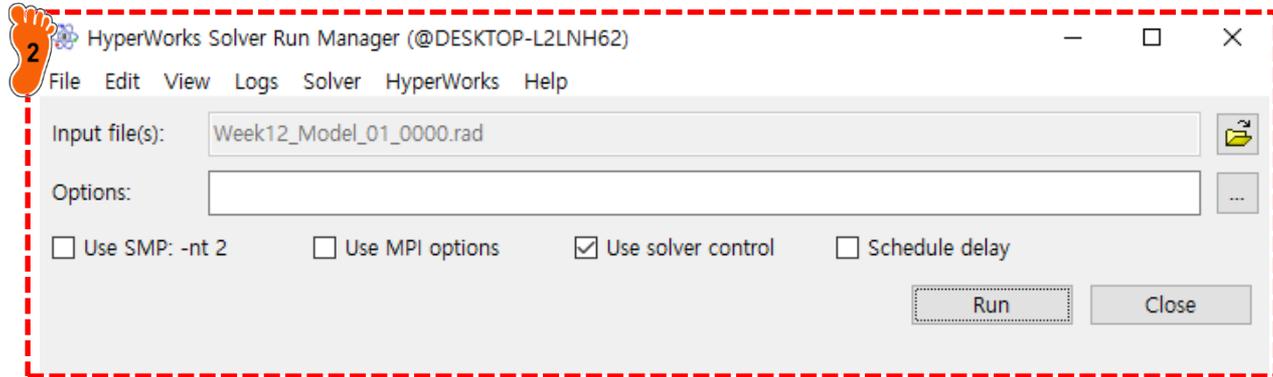
RADIOSS...

Nastran...

.UNV File...

LS-DYNA...

해석 실행 (RADIOSS)



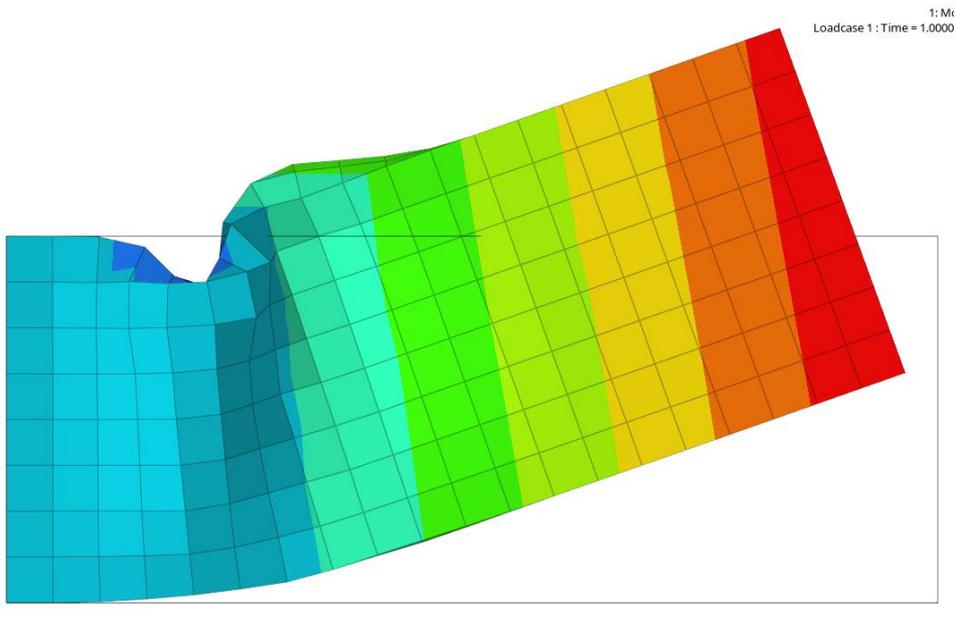
1 Radioss2020 실행

2 앞서 제작한 모델 불러온 뒤 Run 클릭

Contour Plot
Displacement(Z)
Analysis system

5.014E+01
4.284E+01
3.554E+01
2.823E+01
2.093E+01
1.363E+01
6.327E+00
-9.756E-01
-8.278E+00
-1.558E+01
No Result

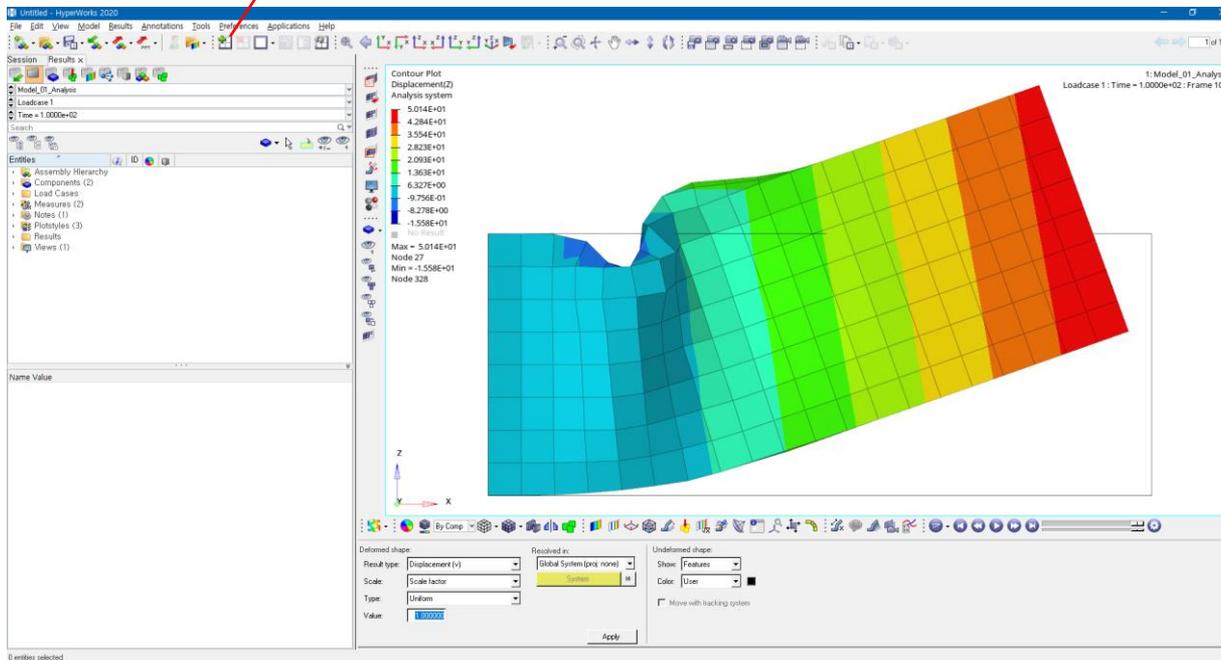
Max = 5.014E+01
Node 27
Min = -1.558E+01
Node 328



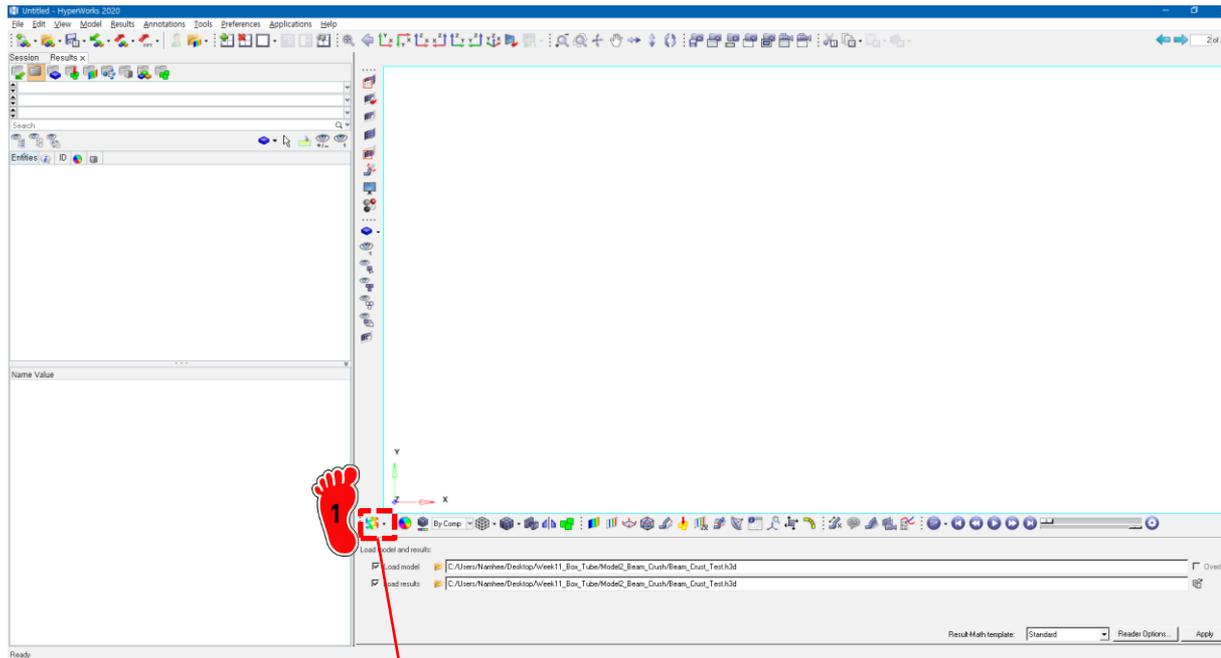
후처리 (1)



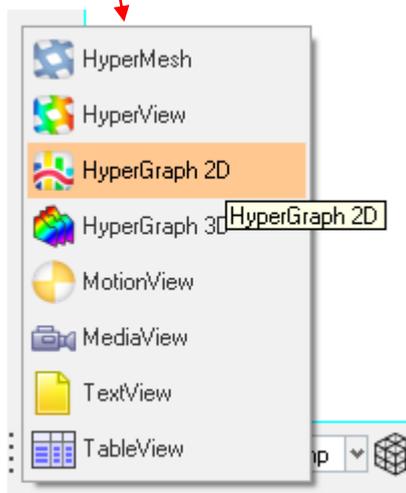
1 HyperView에서 Add Page
버튼 클릭



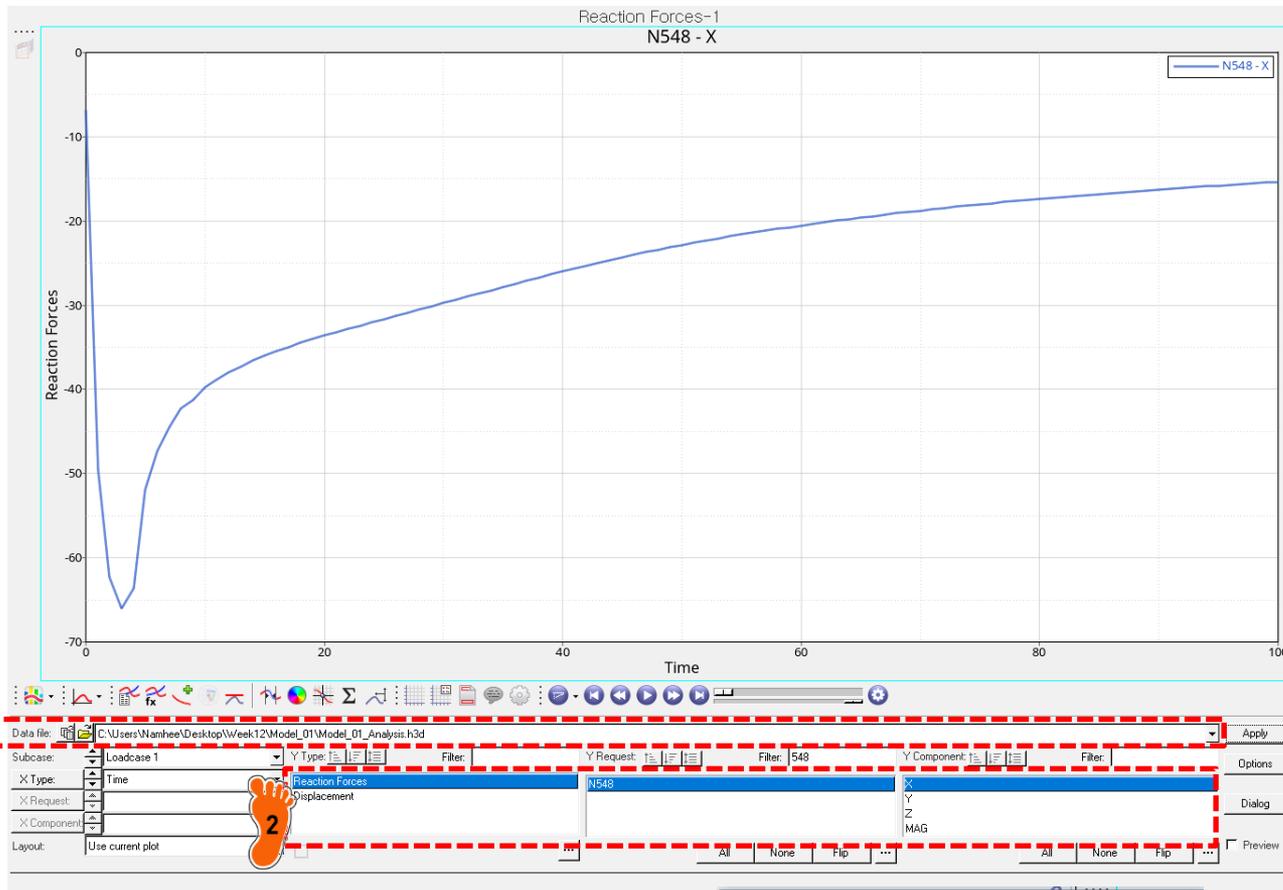
후처리 (2)



1 HyperGraph 2D 클릭



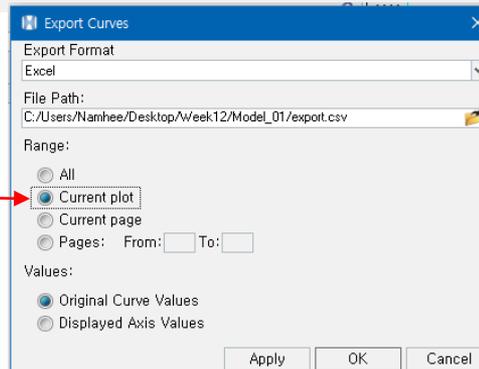
후처리 (3)



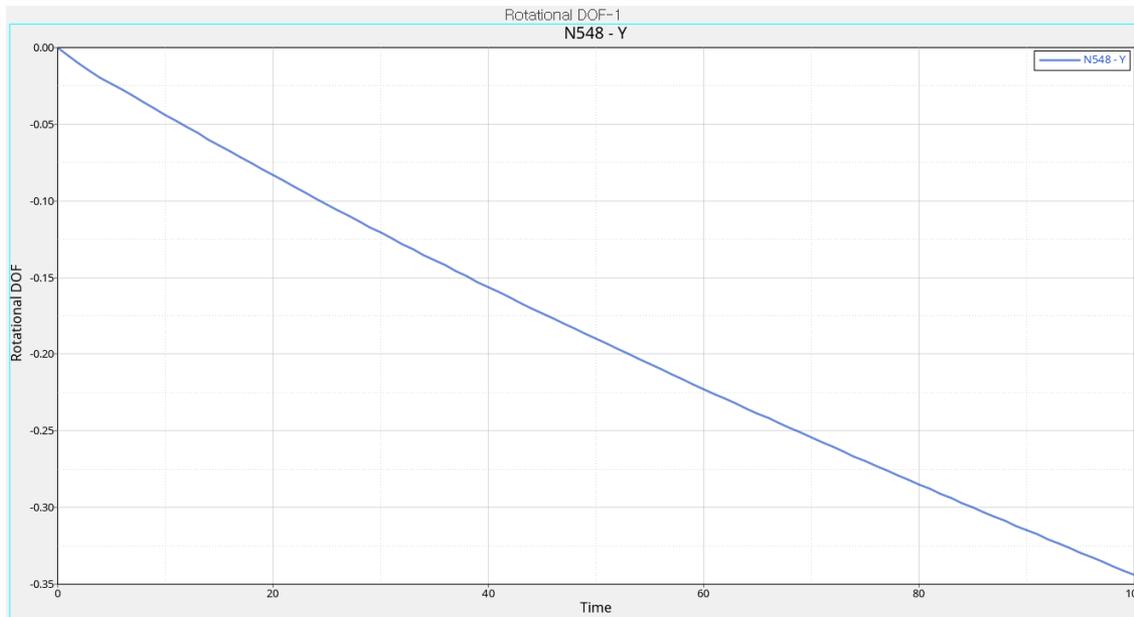
1 해석 결과 파일 불러오기
(*h3d 파일)

2 Reaction Forces 선택,
548번 절점 선택,
X 방향으로의 그래프 확인

3 그래프 csv파일로 저장



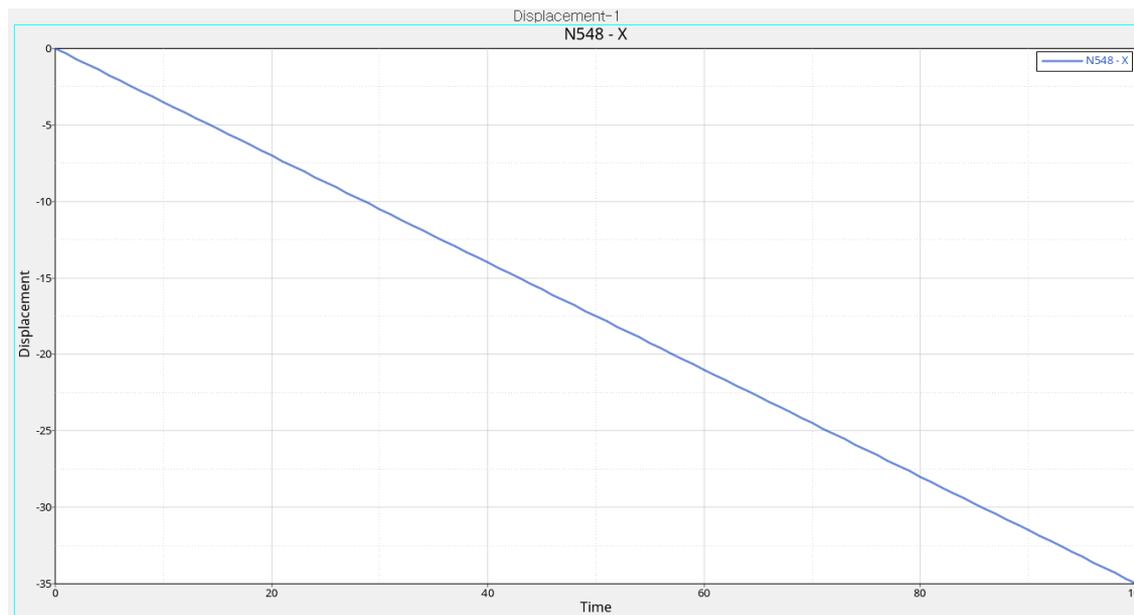
후처리 (4)



새로운 그래프 추가 후
Rotational DOF 선택,
548번 절점 선택,
Y 방향으로의 그래프 확인
→ 그래프 csv파일로 저장



새로운 그래프 추가 후
Displacement 선택,
548번 절점 선택,
X 방향으로의 그래프 확인
→ 그래프 csv파일로 저장



후처리 (5)



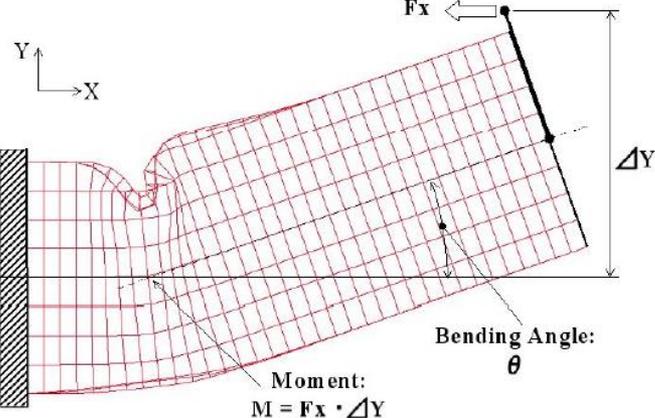
	A	B	C	D	E	F	G	H	I	J
	time	rf	disp x	del y	half	rotation	moment		rotation	moment
2	0.00E+00	-7.36E+00	0.00E+00	0.00E+00	43.30	0.00E+00	-3.19E+02		0.00	318.73
3	1.00E+00	-2.68E+01	-3.50E-01	1.86E-03	43.30	-5.31E-03	-1.16E+03		0.01	1158.81
4	2.00E+00	-5.25E+01	-7.00E-01	7.48E-03	43.30	-1.07E-02	-2.28E+03		0.01	2275.07
5	3.00E+00	-5.96E+01	-1.05E+00	1.67E-02	43.30	-1.59E-02	-2.58E+03		0.02	2583.59
6	4.00E+00	-6.31E+01	-1.40E+00	2.92E-02	43.30	-2.08E-02	-2.73E+03		0.02	2732.08
7	5.00E+00	-5.65E+01	-1.75E+00	4.40E-02	43.30	-2.52E-02	-2.45E+03		0.03	2448.73
8	6.00E+00	-5.04E+01	-2.10E+00	6.16E-02	43.30	-2.93E-02	-2.18E+03		0.03	2184.33
9	7.00E+00	-4.67E+01	-2.45E+00	8.20E-02	43.30	-3.35E-02	-2.03E+03		0.03	2027.62
10	8.00E+00	-4.42E+01	-2.80E+00	1.05E-01	43.30	-3.76E-02	-1.92E+03		0.04	1918.20
11	9.00E+00	-4.23E+01	-3.15E+00	1.31E-01	43.30	-4.17E-02	-1.84E+03		0.04	1836.42
12	1.00E+01	-4.09E+01	-3.50E+00	1.60E-01	43.30	-4.57E-02	-1.78E+03		0.05	1778.37
13	1.10E+01	-3.97E+01	-3.85E+00	1.92E-01	43.30	-4.98E-02	-1.73E+03		0.05	1726.88
14	1.20E+01	-3.88E+01	-4.20E+00	2.26E-01	43.30	-5.38E-02	-1.69E+03		0.05	1687.72
15	1.30E+01	-3.79E+01	-4.55E+00	2.63E-01	43.30	-5.78E-02	-1.65E+03		0.06	1652.18
16	1.40E+01	-3.72E+01	-4.90E+00	3.03E-01	43.30	-6.18E-02	-1.62E+03		0.06	1623.34
17	1.50E+01	-3.66E+01	-5.25E+00	3.45E-01	43.30	-6.57E-02	-1.60E+03		0.07	1598.57
18	1.60E+01	-3.61E+01	-5.60E+00	3.90E-01	43.30	-6.97E-02	-1.58E+03		0.07	1576.58



Excel을 통해 Moment 계산

Reaction force:

F_x

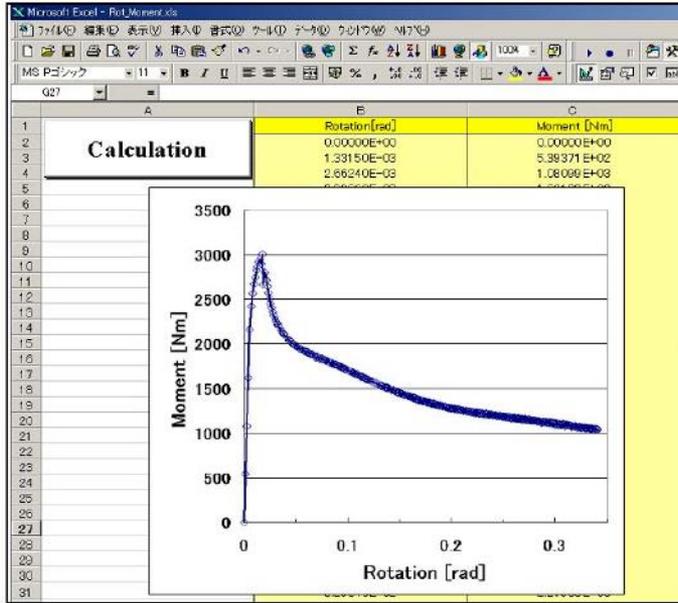


$$\nabla Y = (\text{displacement } X) \cdot (\text{rotation})$$

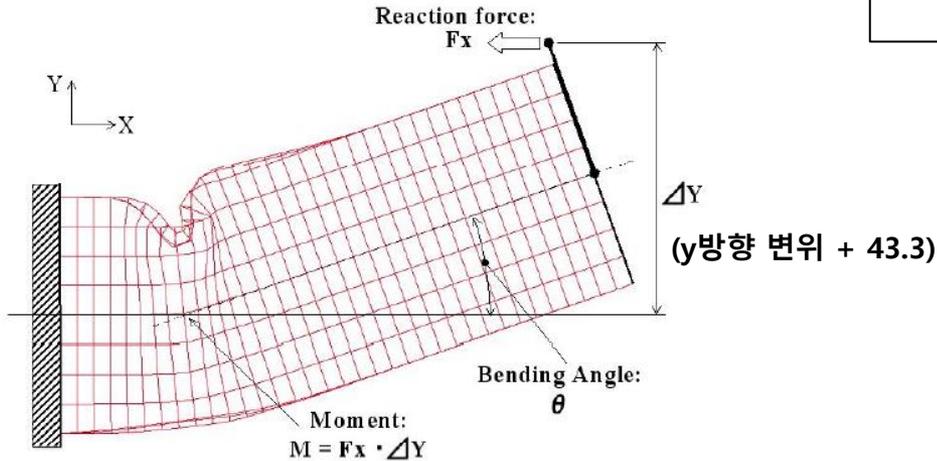
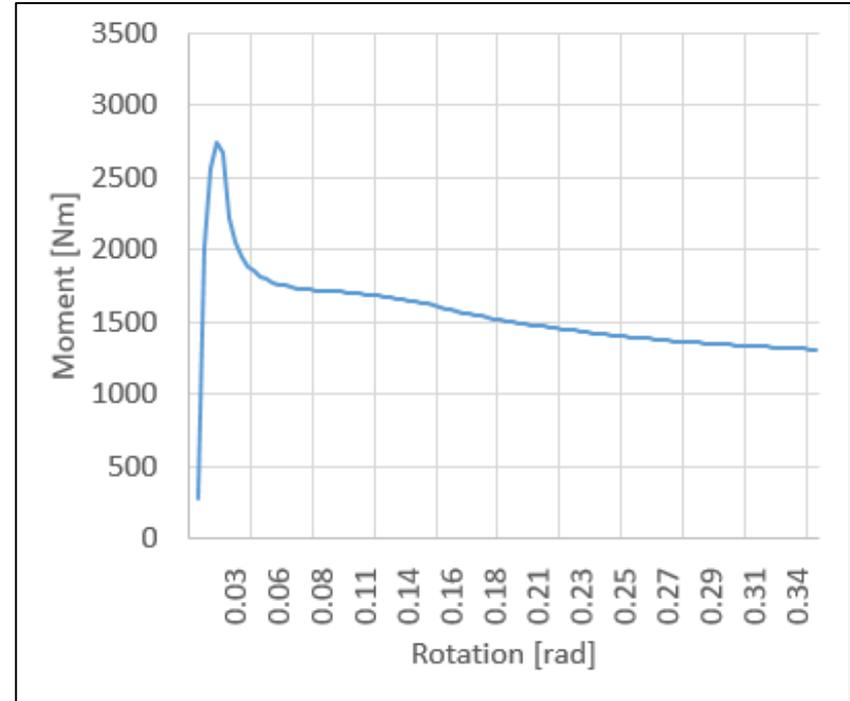
$$\text{Moment} = (\text{reaction force}) \cdot (\text{del } Y + 43.3)$$

후처리 (5)

참고 논문 결과

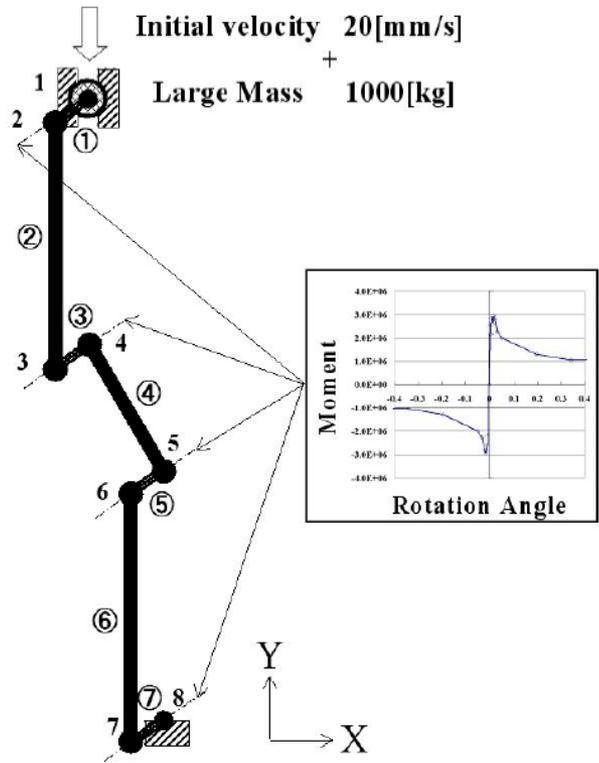


해석 결과



MID RAIL ANALYSIS 1 (BEAM ELEMENT)

예제: JOINT MOMENT ANALYSIS



Microsoft Excel - Trial_F0A_v1001.xls

1. Input Data into below Sheet 2. Modify Cross-section / B.C. 3D-View 3. Produce Beam model LS-I

Nodes	Elements	Load, Constr.	L+ Value, (Example L10)	Yez = 1
8	7			

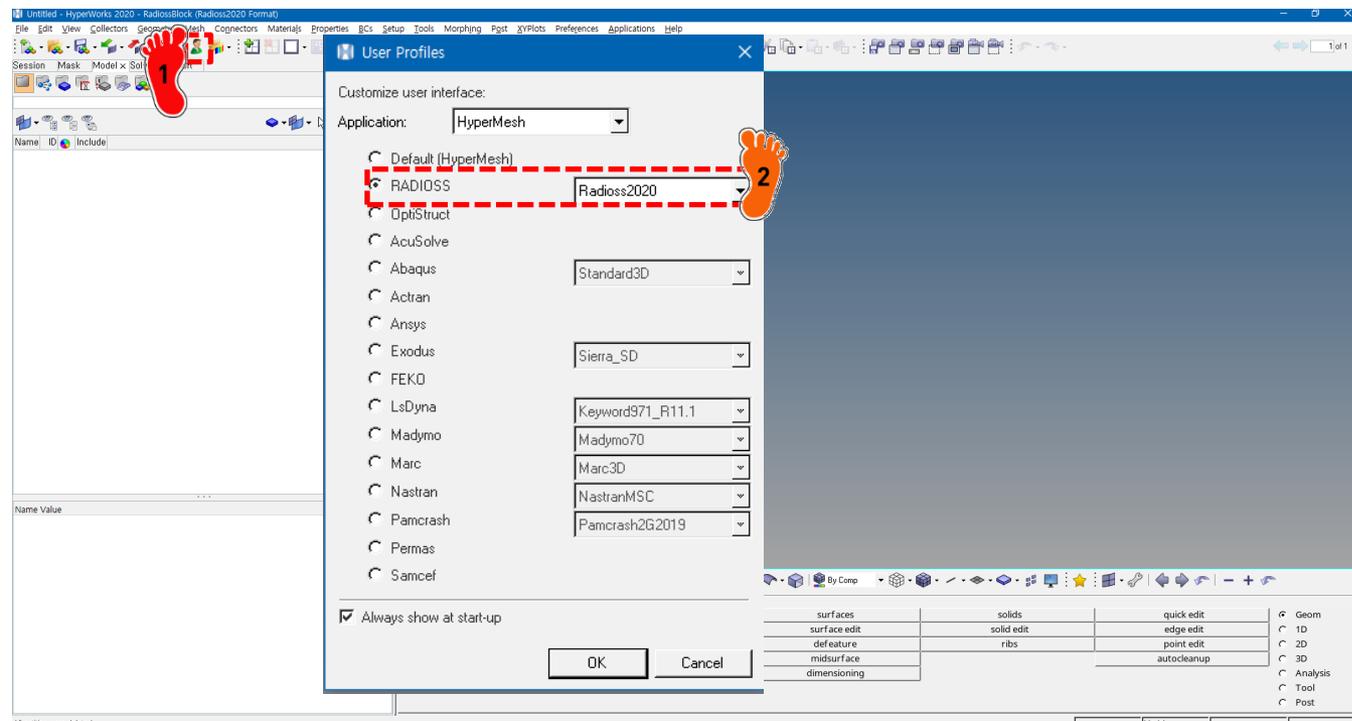
Node No.	X [mm]	Y [mm]	Z [mm]	X	Y	Z	rot-X	rot-Y	rot-Z	Elem. No.	Node1	Node2	E[N/mm2]	ν	ρ [kg/mm3]
1	0.00000E+00	1.00000E+08	0.00000E+00	1	0	0	1	1	1	1	1	2	208000	0.3	7.85E-06
2	0.00000E+00	1.00000E+08	1.00000E+00	1	1	1	1	1	1	2	3	3	208000	0.3	7.85E-06
3	0.00000E+00	8.00000E+02	1.00000E+00	1	1	1	1	1	1	3	4	4	208000	0.3	7.85E-06
4	0.00000E+00	6.00000E+02	0.00000E+00	1	1	1	1	1	1	4	5	5	208000	0.3	7.85E-06
5	1.50000E+02	4.00000E+02	0.00000E+00	1	1	1	1	1	1	5	6	6	208000	0.3	7.85E-06
6	1.50000E+02	4.00000E+02	1.00000E+00	1	1	1	1	1	1	6	7	7	208000	0.3	7.85E-06
7	1.50000E+02	0.00000E+00	1.00000E+00	1	1	1	1	1	1	7	8	8	208000	0.3	7.85E-06
8	1.50000E+02	0.00000E+00	0.00000E+00	1	1	1	1	1	1	8					

Total Masses= 3.44194E+00

Click!

END

기하형상 생성 (1)



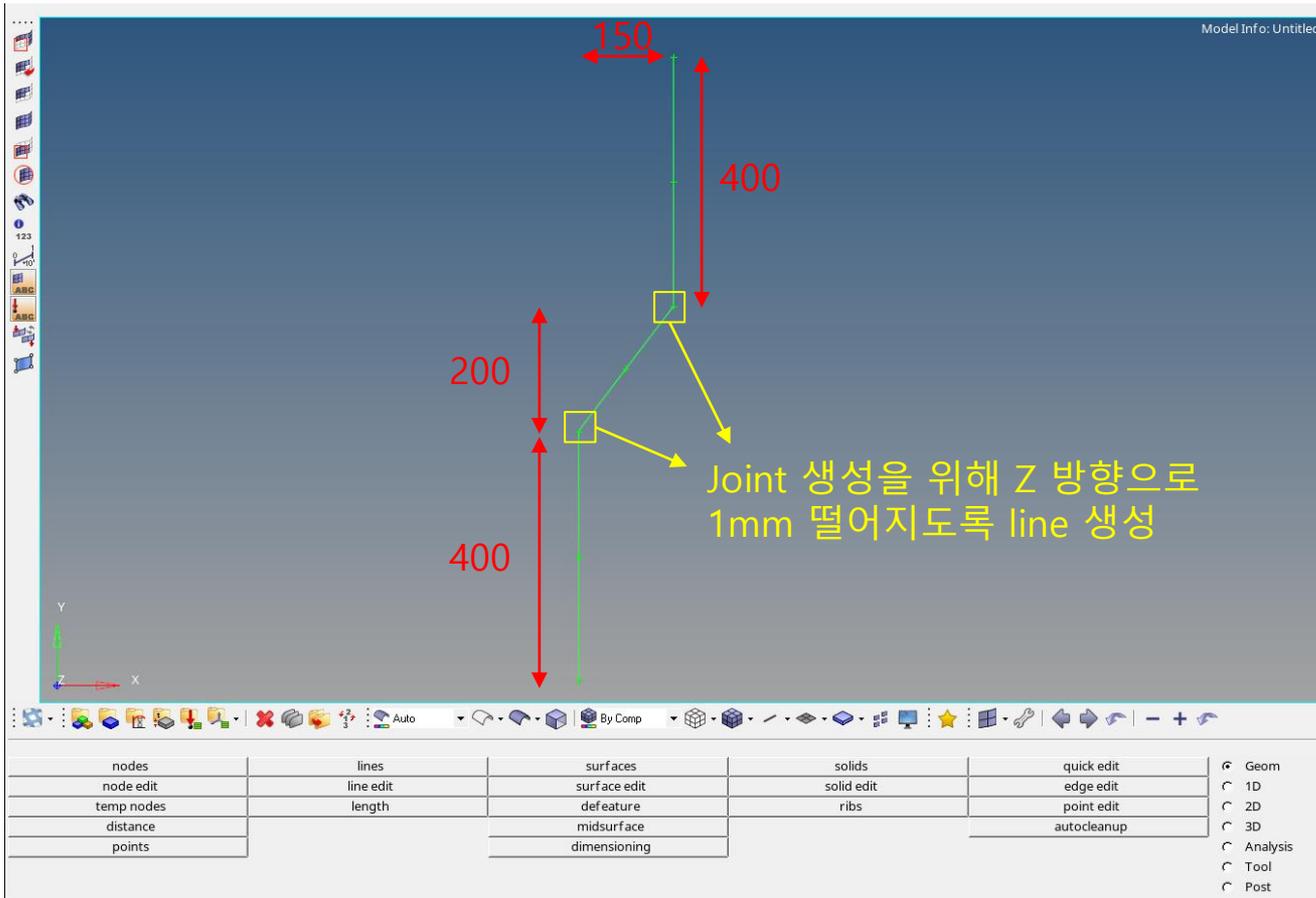
1 User Profiles () 클릭

2 RADIOSS 선택

기하형상 생성 (2)



3개의 line 생성



$(0,0,0) - (0,400,0)$
 $(0,400,1) - (150,600,1)$
 $(150,600,0) - (150, 1000, 0)$

재료 입력

Components (1)			
auto1	1	<input checked="" type="checkbox"/>	0
Materials (1)			
material1	1	<input checked="" type="checkbox"/>	0
Titles (1)			
Model Info	1	<input type="checkbox"/>	0

Name	Value
Solver Keyword	/MAT/ELAST/
ID	1
Name	material1
Color	<input checked="" type="checkbox"/>
Include	[Master Model]
Defined	<input checked="" type="checkbox"/>
User Comments	Hide In Menu/Export
Card Image	M1_ELAST
Regular_OR_encrypted_flag	Regular
RefRho_Option	<input type="checkbox"/>
Rho_Initial	7,85e-09
E	206000,0
Nu	0,3



선형 재료 생성

Card Image: M1_ELAST

Rho_Initial: 7.85e-9

E: 206 GPa

Nu: 0.3

요소망 생성 (1)

1

ID	1
Name	beamsection1
Include	[Master Model]
Collector	(1) beamsectcol1
Config	Standard
Section Type	HMBOX
Standard angle	0,0
Parameter Definitions	
Dimension a	50,0
Dimension b	80,0
Thickness t	1,6

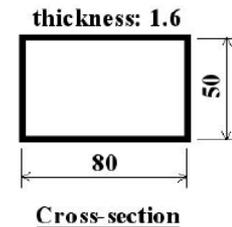
2

Name	Value
Solver Keyword	/PROP/BEAM/
ID	1
Name	property1
Color	
Include	[Master Model]
Defined	<input checked="" type="checkbox"/>
User Comments	Hide In Menu/Export
Card Image	P3_BEAM
Hyperbeamsection	(1) beamsection1
Regular_OR_encrypted_flag	Regular

3

Name	Value
Solver Keyword	/PART/
ID	1
Name	auto1
Color	
Include	[Master Model]
Defined	<input checked="" type="checkbox"/>
User Comments	Hide In Menu/Export
Card Image	Part
Prop_Id	(1) property1
Mat_Id	(1) material1

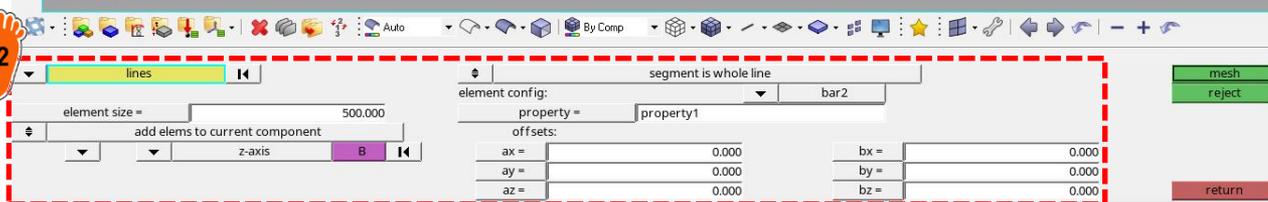
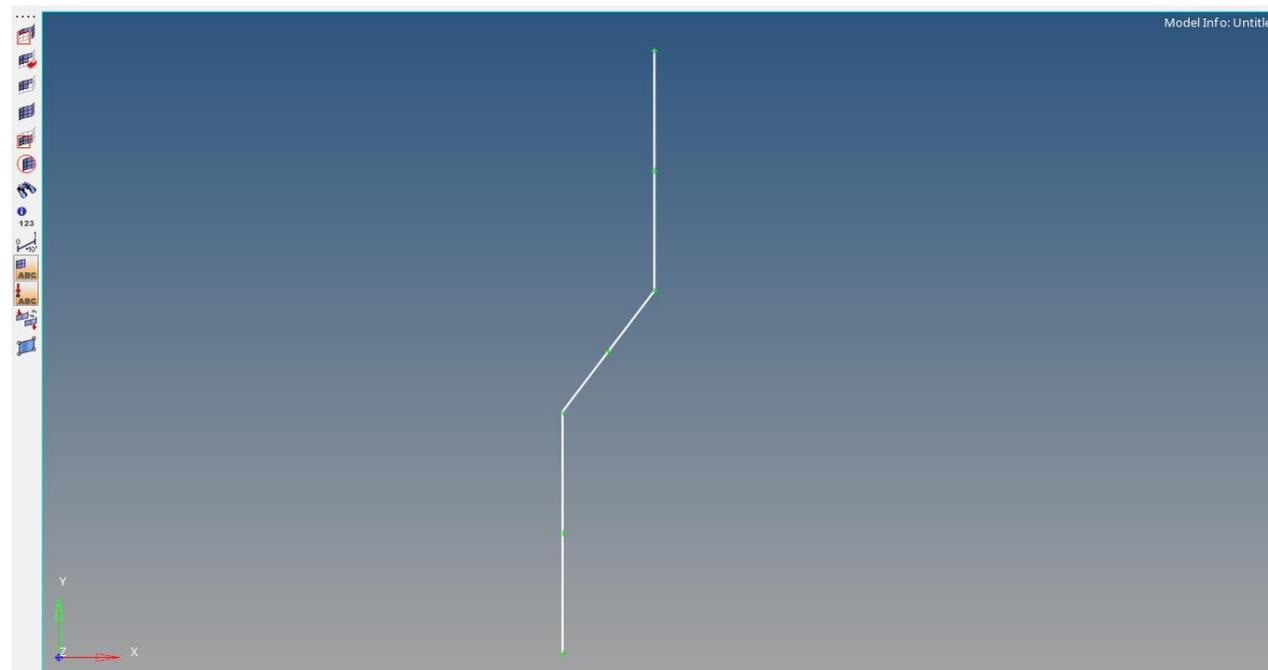
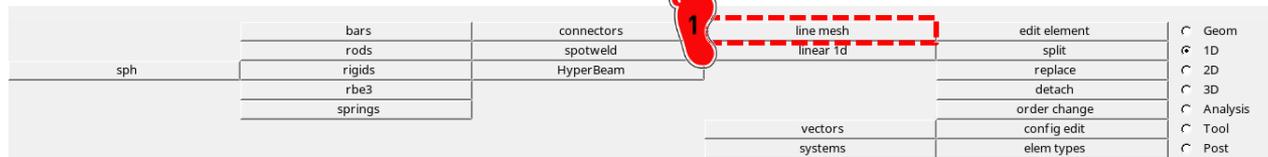
1 Beam section 생성



2 Property 생성
Card Image: P3_BEAM
Beam section 설정

3 Component에 특성, 재료
설정
Card Image는 Part로 변경

요소망 생성 (2)



1 1D > line mesh

2 lines 선택
 element size : 500 입력
 element config: bar2 선택
 property 설정
 Orientation: z-axis 설정

조인트 요소 생성 (1)

1

Name	Value
Solver Keyword	/PROP/SPR_GENE/
ID	2
Name	property2
Color	
Include	[Master Model]
Defined	<input checked="" type="checkbox"/>
User Comments	Hide In Menu/Export
Card Image	P8_SPR_GENE
Regular_OR_encrypted_flag	Regular
MASS	0,0001
Inertia	0,0001

Prop/Type8 (SPR_GENE)
: General spring property is used to model a joint connection between two parts with six independent modes of deformation.

1 Joint property 생성
Card Image: P8_SPR_GENE

Mass: 0.0001
Inertia: 0.0001

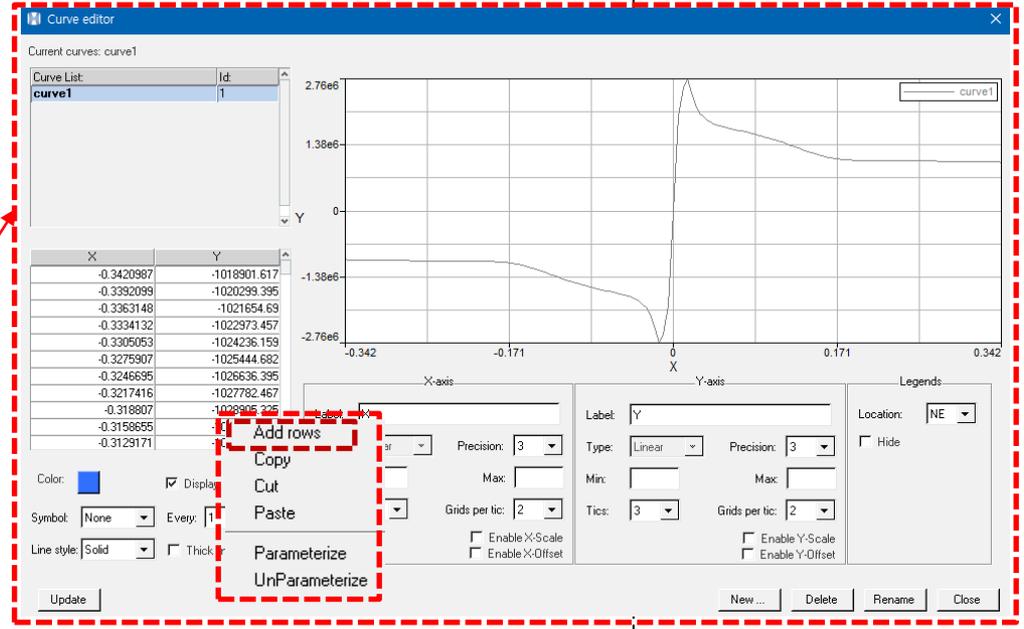
Tx,Ty,Tz,Rx,Ry:
K_TensX 등 = 1e10 (K1~5)

2

Name	Value
Rotational(Rz)	
K_TorsZ	0.0
C_TorsZ	
A_TorsZ	
B_TorsZ	
D_TorsZ	
fct_IDi7	<Unspecified>
H_TorsZ	
fct_IDi8	
fct_IDi9	
fct_IDrz4	
ThetaMin_TorsZ	
ThetaMax_TorsZ	
F_TorsZ	
E_TorsZ	
Ascale_TorsZ	
H_TorsZ	

2 Rz: K_TorsZ = 0 (K6),
fct_IDi7(rotation in Z)
활성화하여 회전 강성 테이블
입력

3



조인트 요소 생성 (2)

Name	ID	Include
Beam Section Collectors (1)		
beamsectcoll	1	0
beamsection1	1	0
Components (2)		
auto1	1	0
Joint	2	0

Name	Value
Solver Keyword	/PART/
ID	2
Name	Joint
Color	■
Include	[Master Model]
Defined	<input checked="" type="checkbox"/>
User Comments	Hide In Menu/Export
Card Image	Part
Prop_Id	(2) property2
Mat_Id	<Unspecified>

1 Joint component 생성 후 property 부여

2 1D>spring

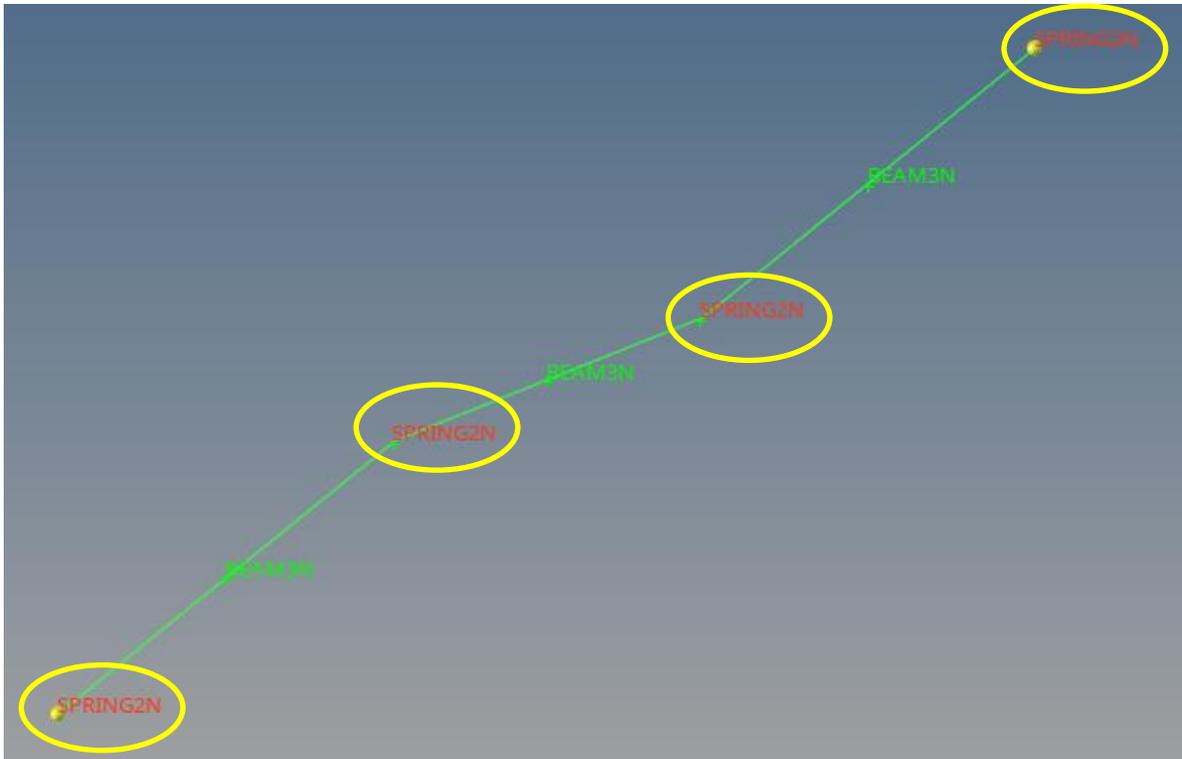
3 spring2 생성에서 절점 두 개 선택하여 joint 요소 생성

	bars	connectors	line mesh	edit element	<input type="checkbox"/> Geom
	rods	spotweld	linear 1d	split	<input checked="" type="checkbox"/> 1D
sph	rigids	HyperBeam		replace	<input type="checkbox"/> 2D
	rbe3			detach	<input type="checkbox"/> 3D
	springs			order change	<input type="checkbox"/> Analysis
			vectors	config edit	<input type="checkbox"/> Tool
			systems	elem types	<input type="checkbox"/> Post

spring2	node	reject
spring3	node	review
spring4		

return

조인트 요소 생성 (3)

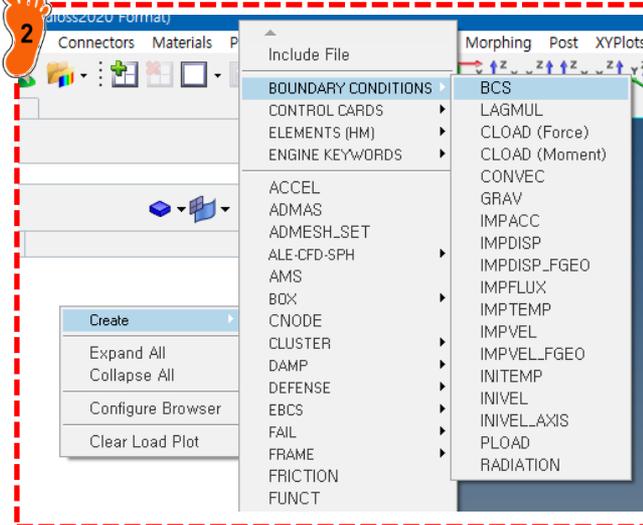
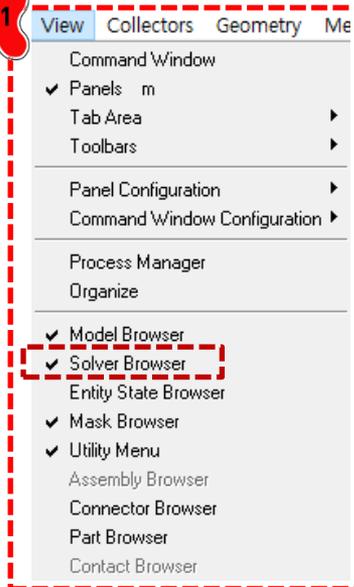


1 네 군데 joint 요소 생성

2 양 끝단에는 temp node를 생성하여 joint를 생성함

Temp node: (0,0,1),
(150, 1000, 1)

구속조건 설정 (1)



View>Solver Browser 체크

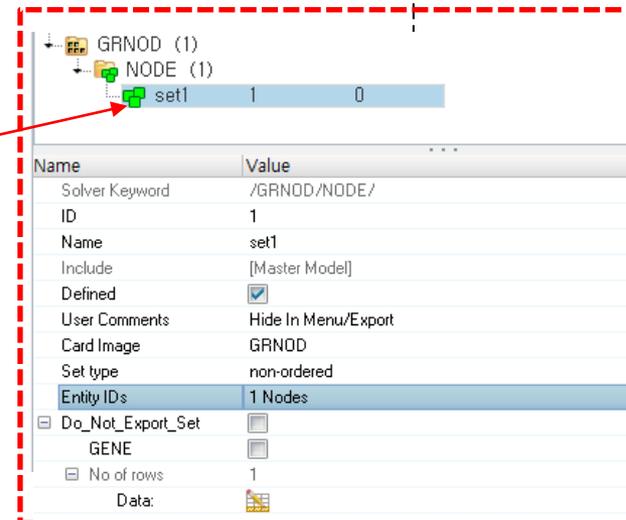
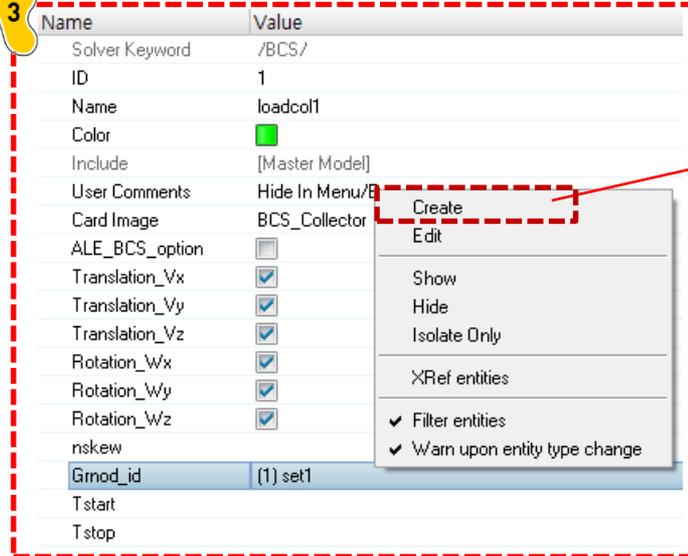


Solver 탭에서 마우스 우클릭 하여 BCS 생성

DOF 1~6 모두 구속

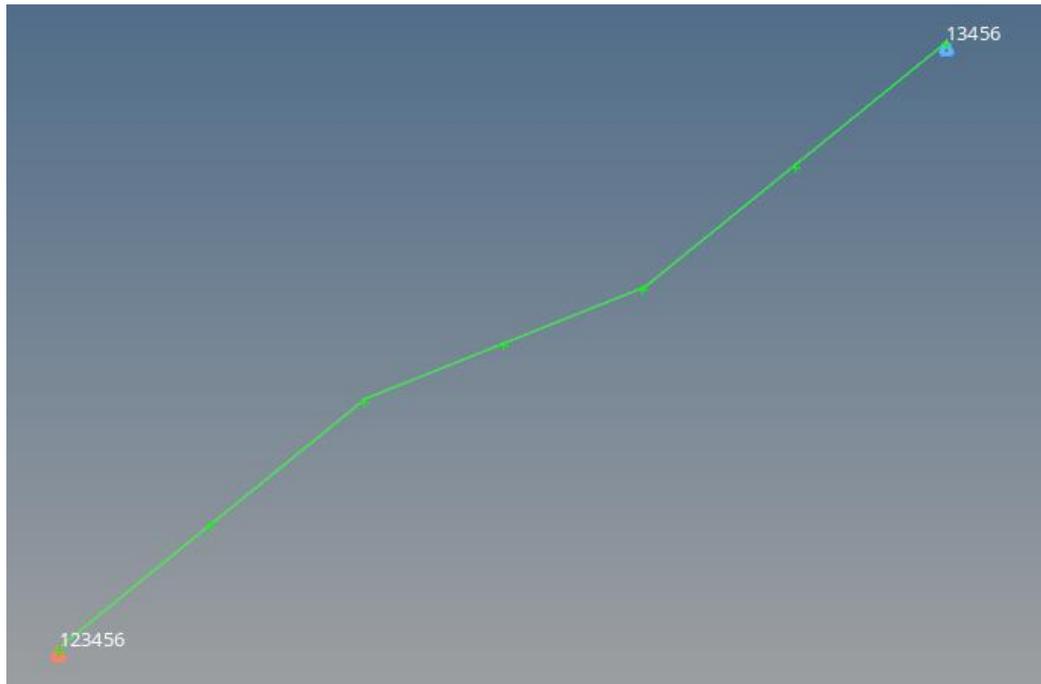


grnod_id 우클릭 하여 절점 set 생성
→ 고정 구속할 노드 추가



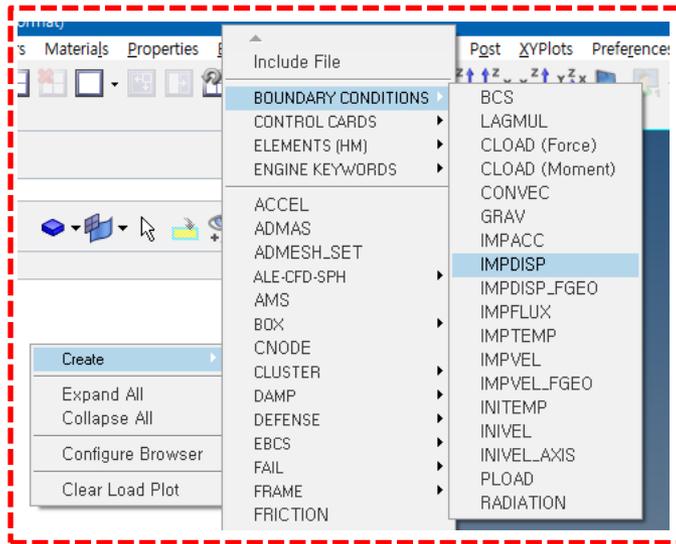
구속조건 설정 (2)

Name	Value
Solver Keyword	/BCS/
ID	2
Name	loadcol2
Color	■
Include	[Master Model]
User Comments	Hide In Menu/Export
Card Image	BCS_Collector
ALE_BCS_option	<input type="checkbox"/>
Translation_Vx	<input checked="" type="checkbox"/>
Translation_Vy	<input type="checkbox"/>
Translation_Vz	<input checked="" type="checkbox"/>
Rotation_Wx	<input checked="" type="checkbox"/>
Rotation_Wy	<input checked="" type="checkbox"/>
Rotation_Wz	<input checked="" type="checkbox"/>



같은 방법으로 반대편
절점은 DOF2 제외하고
모두 구속

변위 조건 설정 (1)

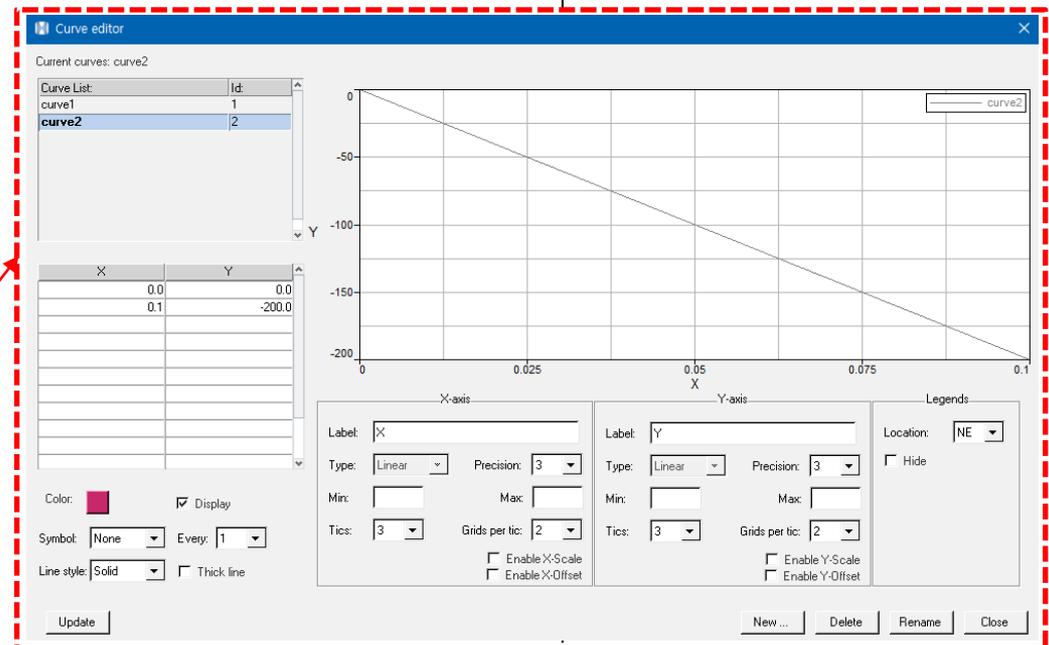


1 Solver탭 우클릭하여 IMPDISP 생성

2 Ifunct 우클릭 > Create curve2 우클릭 > Edit

시간-변위 함수 생성
(0,0) (0.1,-200)

Name	Value
Solver Keyword	/IMPDISP/
ID	3
Name	loadcol3
Color	1
Include	[Master Model]
User Comments	Hide In Menu/Export
Card Image	IMPDISP_Collector
Ifunct	
DIR	X
Input Type	SKEW
Iskew	



변위 조건 설정 (2)

Name	Value
Solver Keyword	/IMPDISP/
ID	3
Name	loadcol3
Color	
Include	[Master Model]
User Comments	Hide In Menu/Export
Card Image	IMPDISP_Collector
Ifunct	[2] curve2
DIR	Y
Input Type	SKEW
lskew	
lsensor	
Gmod_id	
l_coor	
scale_x	
scale_y	
Tstart	
Tstop	



Create

Edit

Show

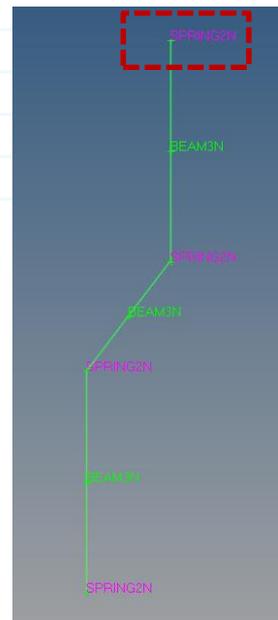
Hide

Isolate Only

XRef entities

Filter entities

Warn upon entity type change



1 Grnod_id 클릭하여 set
생성 후 절점 추가
(상단 노드)



2 Dir은 Y로 설정

해석 케이스 설정

1

ENGINE_KEYWORDS (1)

- RUN (1)
 - RUN (1)
 - ENG_RUN 1 0

Name	Value
Solver Keyword	/RUN///
Include	[Master Model]
Name	ENG_RUN
Status	<input checked="" type="checkbox"/>
RunName	
RunNumber	1
RestartLetter	
Tstop	0.1

2

ANIM (1)

- ANIM/DT (1)
 - ENG_ANIM_DT 2 0

Name	Value
Solver Keyword	/ANIM/DT
Include	[Master Model]
Name	ENG_ANIM_DT
Status	<input checked="" type="checkbox"/>
Tstart	0.0
Tfreq	0.001

3

ANIM/VECT (1)

- ENG_ANIM_VECT 3 0

Name	Value
Solver Keyword	/ANIM/VECT/FREAC
Include	[Master Model]
Name	ENG_ANIM_VECT
Status	<input checked="" type="checkbox"/>
ACC	<input type="checkbox"/>
CONT	<input type="checkbox"/>
DISP	<input type="checkbox"/>
FINT	<input type="checkbox"/>
FVEL	<input type="checkbox"/>
PCONT	<input type="checkbox"/>
VEL	<input type="checkbox"/>
FEXT	<input type="checkbox"/>
CONT2	<input type="checkbox"/>
FOPT	<input type="checkbox"/>
VROT	<input type="checkbox"/>
DROT	<input type="checkbox"/>
VFLU	<input type="checkbox"/>
FREAC	<input checked="" type="checkbox"/>
MREAC	<input type="checkbox"/>
CLUSTER	<input type="checkbox"/>

: Reaction forces for imposed velocities, displacements, accelerations and boundary conditions.

우클릭>Create>
ENGINE_KEYWORDS

1 [ENG_RUN]
RunNumber: 1
Tstop: 0.1

2 [ENG_ANIM_DT]
Tfreq: 0.001

3 [ENG_ANIM_VECT]
FREAC

해석 실행



File selection

File type: RADIOSS

Template: Radioss2017

File: cuments\hw_practice\12\new1\FDA_joint_1D_beam_0000.rad

Export options

Export: Custom

Solver options:



HyperWorks Solver Run Manager (@DESKTOP-L2LNH62)

File Edit View Logs Solver HyperWorks Help

Input file(s): FOA_joint_1D_beam_0000.rad

Options:

Use SMP: -nt 2 Use MPI options Use solver control Schedule delay



모델 export



Radioss에서 해석 실행

후처리



변형된 형상 확인

참고 논문 결과

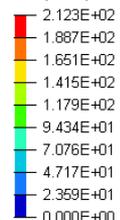


Initial

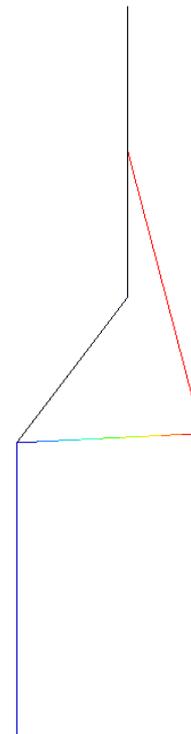
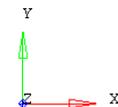
Deformed

해석 결과

Contour Plot
Displacement(Mag)
Analysis system



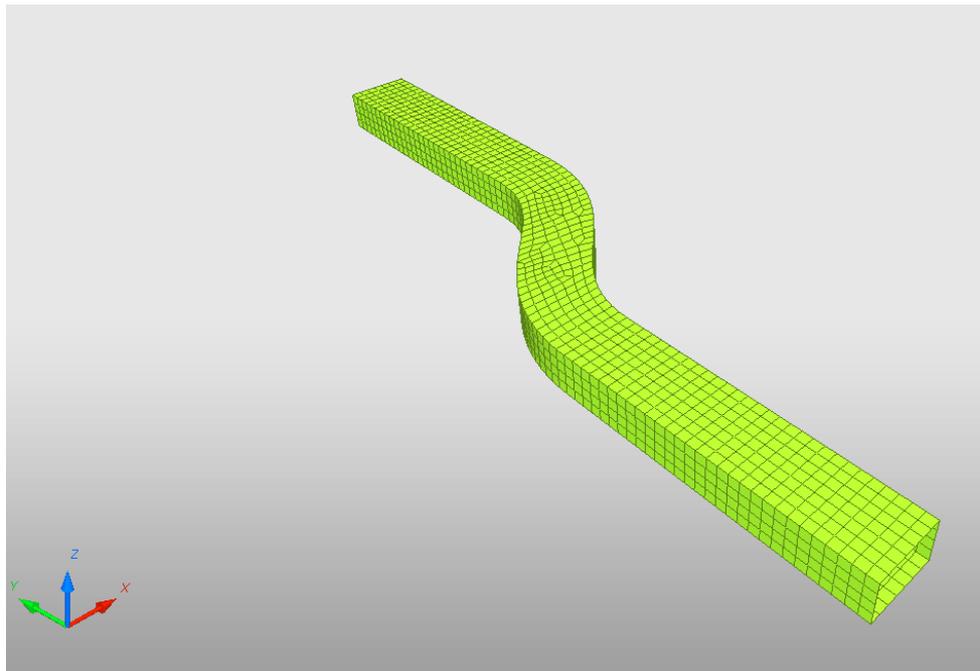
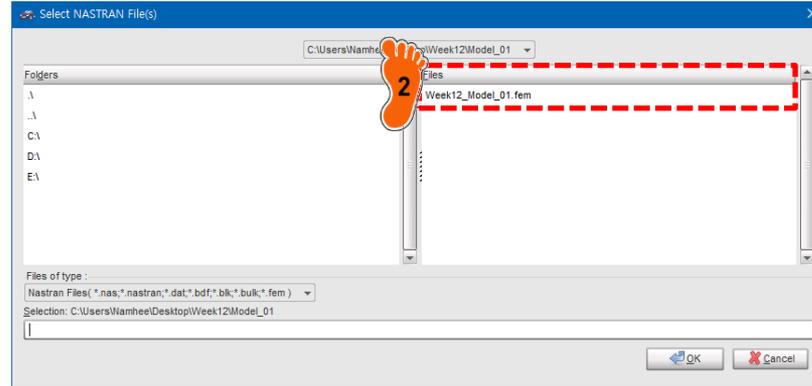
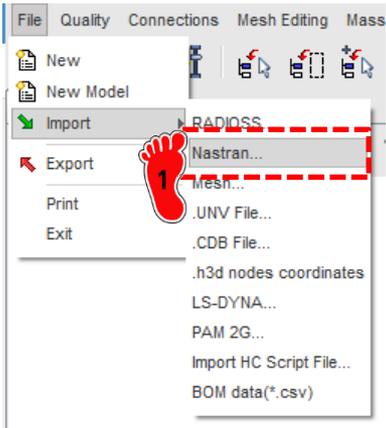
■ No result
Max = 2.123E+02
Node 4
Min = 0.000E+00
Node 7



Computational time: 78.34 s

MID RAIL ANALYSIS 2 (SHELL ELEMENT)

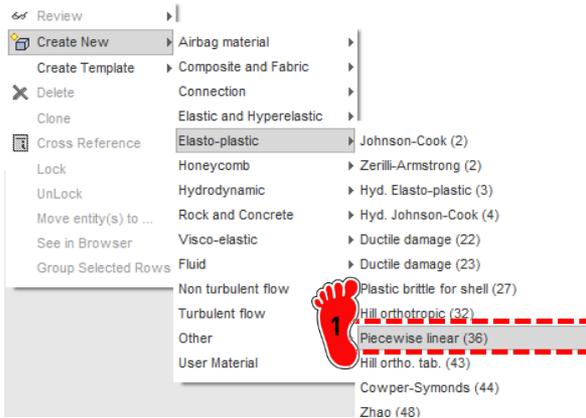
모델 불러오기



1 File > Import > Nastran
클릭

2 모델 파일 클릭 후 불러오기
(Week12_Model_02.fem)

재료 물성 입력 (1)



1 Model > Material 클릭,
마우스 우클릭 후
Create New >
Piecewise linear(36) 재료
생성

2 재료 물성치 입력
[RHO_I]: 7.85e-6
[E]: 240
[Nu]: 0.3

PLAS_TAB	
ID	2
Title	New MAT 2
Local Unit System	None
[RHO_I] Initial density*	7.85E-6
[RHO_0] Reference density	0
[E] Young's modulus*	206
[Nu] Poisson's ratio*	.3
[Eps_p_max] Failure plastic strain	0
[Eps_t] Tensile failure strain	0
[Eps_m] Tensile failure strain	0
[C_hard] Hardening coefficient	0
[F_smooth] Strain rate smoothing	<input type="checkbox"/>
[F_cut] Cutoff frequency for strain rate filtering	0
[Eps_f] Maximum tensile failure strain	0
[VP] Strain rate choice flag	0: Strain rate are total strain rate
[fct_IDp] Pressure vs. yield factor function	None
[Fscale] Y-Scale factor for yield factor function	1
[Fct_IDE] Function identifier for the scale factor of You	None
[EInf] Saturated Young's modulus for infinitive plastic s	0
[CE] Parameter for Young's modulus evolution	0
Heat	<input type="checkbox"/>

재료 물성 입력 (2)

1 Stress-Strain curve 입력
(마우스 우클릭 후 "Define function" 클릭)

2 "Import curve" 클릭 후
"Material Property.txt" 파일 불러오기

3 모델에 재료 적용

PLAS_TAB

[Eps_m] Tensile failure strain	0
[C_hard] Hardening coefficient	0
[F_smooth] Strain rate smoothing	<input type="checkbox"/>
[F_cut] Cutoff frequency for strain rate filtering	0
[Eps_f] Maximum tensile failure strain	0
[VP] Strain rate choice flag	0: Strain rate are total strain rate
[fct_Dp] Pressure vs. yield factor function	None
[Fscale] Y-Scale factor for yield factor function	1
[Fct_DE] Function identifier for the scale factor of Young mod	None
[Ein_f] Saturated Young's modulus for infinitive plastic strain	0
[CE] Parameter for Young's modulus evolution	0
Heat	<input type="checkbox"/>

Strain rate

S.No	Eps_dot_1	Fscale_1	func_ID1
1	0	0	[1] New FUNCT 1

[Fscale_1] Scale factor ordinate (stress) [1] New FUNCT 1
 [func_ID1] Yield stress function* [1] New FUNCT 1
 [Eps_dot_1] Strain rate [0] Items

Function Window

Function name: New FUNCT 1

f(x)

1.00E+000
5.00E-001
0.00E+000

0.00E+000 5.00E-001 1.00E+000

x

x: 0
f(x): 0

Validate

Function file

C:\Users\Namhee\Desktop\Week12Model_01

Folders

- \
- ..
- Report_Week12_Model_01
- C:\
- D:\
- E:\

Files

- Material_Property.txt
- Week12_Model_01.fem
- Week12_Model_01.html

Recent Files :

C:\Users\Namhee\Desktop\Week12Model_01\Week12_Model_01.fem

Selection: C:\Users\Namhee\Desktop\Week12Model_01

OK Cancel

Function Window

Function name: New FUNCT 1

f(x)

5.790000E-001
2.895000E-001
0.000000E+000

0.000000E+000 1.500000E-001 3.000000E-001

x

x: 0
f(x): 0

Validate

Undo Reference Save Cancel

특성 입력

SHELL	
ID	2
Title	New PROP 2
Local Unit System	None
[lsmstr] Flag for shell small strain formulation	0: Use value in /DEF_SHELL
[lshell] Flag for 4 node shell element formulation	0: Use value in /DEF_SHELL
[lsh3n] Flag for 3 node shell element formulation	0: use value in /DEF_SHELL
[ldrill] Flag for drilling degree of freedom stiffness	0: No
[P_thick_fail] Percentage of through thickness integration	0
[hm] Shell membrane hourglass coefficient	0
[hf] Shell out of plane hourglass	0
[hr] Shell rotation hourglass coefficient	0
[dm] Shell membrane damping	0
[dn] Shell numerical damping	0
[N] Number of integration points through the thickness	0
[lstrain] Flag to compute strains for post-processing	0: Use value in /DEF_SHELL
[Thick] Shell thickness*	1.6
[Ashear] Shear factor	0
[thick] Flag for shell resultant stresses calculation	0: Default set to value defined with /DEF_SHELL
[lplas] Flag for shell plane stress plasticity	0: Default set to value defined with /DEF_SHELL
Support*	1 items

Close Save Cancel



Model > Property 클릭,
마우스 우클릭 후
Create New > Surface >
Shell (1) 클릭

두께 1.6mm 입력



모델에 특성 적용

RBE2 생성

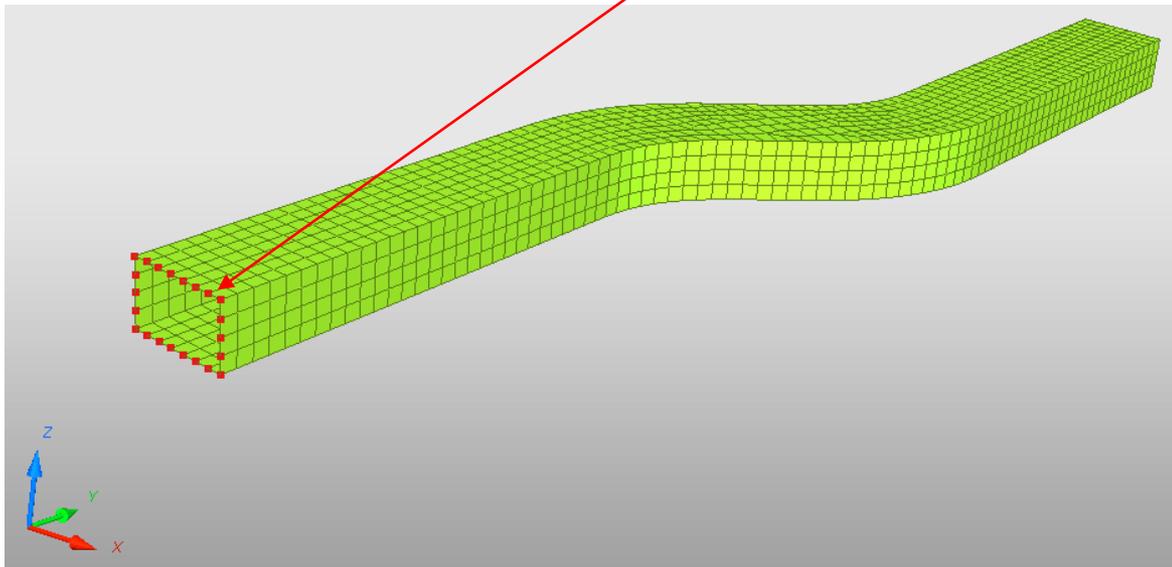
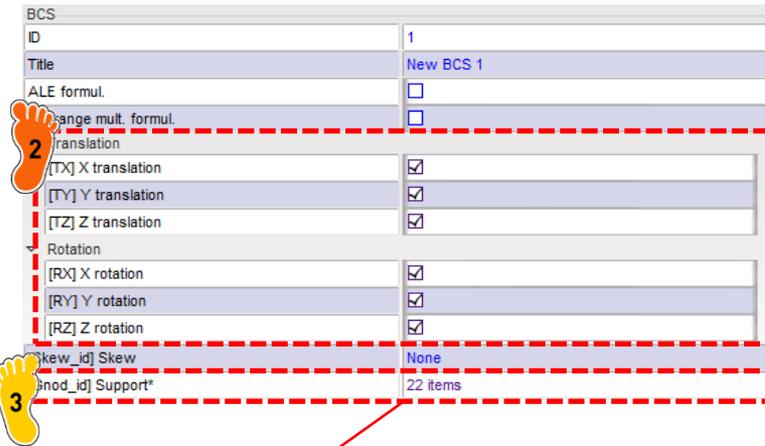
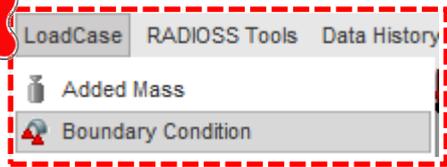
The image illustrates the steps to create an RBE2 constraint in a finite element software. It shows the menu navigation, the RBE2 dialog box configuration, the NODE dialog box for specifying the master node coordinates, and the resulting 3D mesh model with the master node highlighted.

1 Mesh Editing > RBE2 생성
마우스 우클릭 후 Create
New 클릭

2 Master node 좌표:
(150, 1000, 0)

3 Support: Y방향 끝단 노드
선택

구속조건 설정 (1)



LoadCase > Boundary Condition 클릭, 마우스 우클릭 후 Create new 클릭



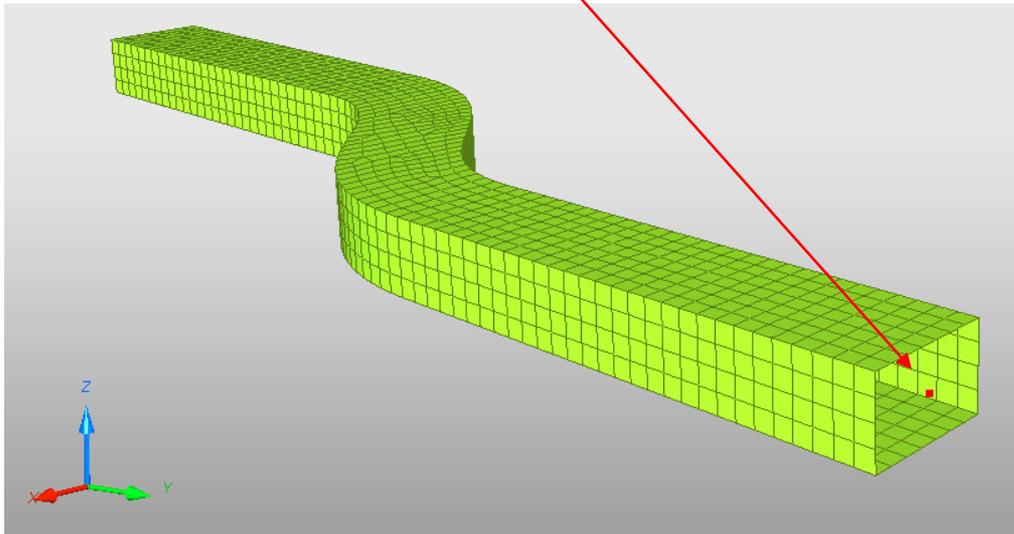
완전 구속 설정



구속할 절점 선택

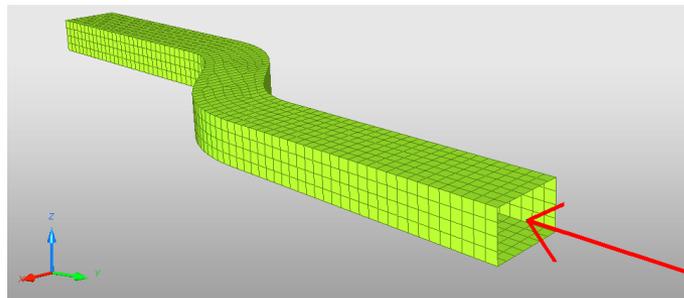
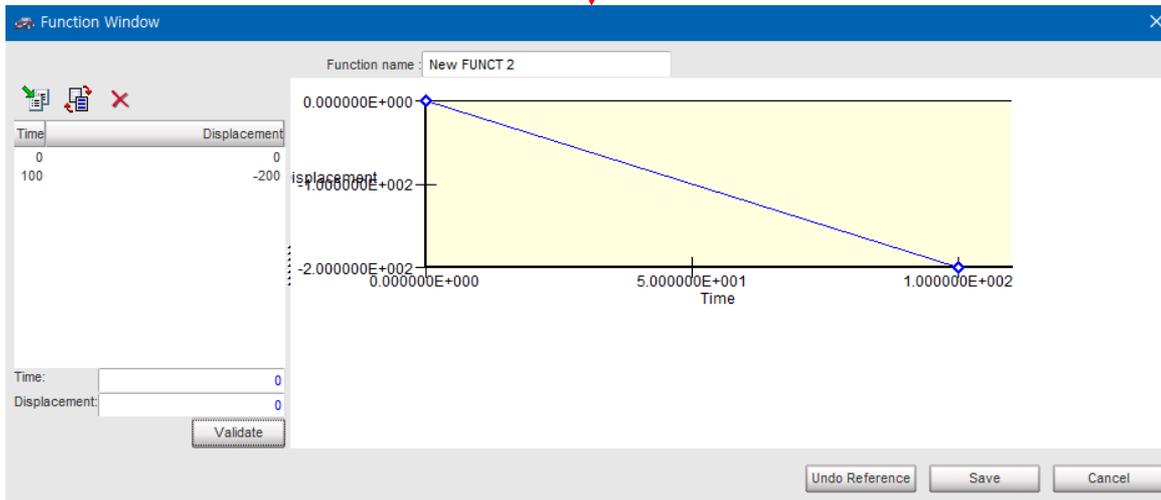
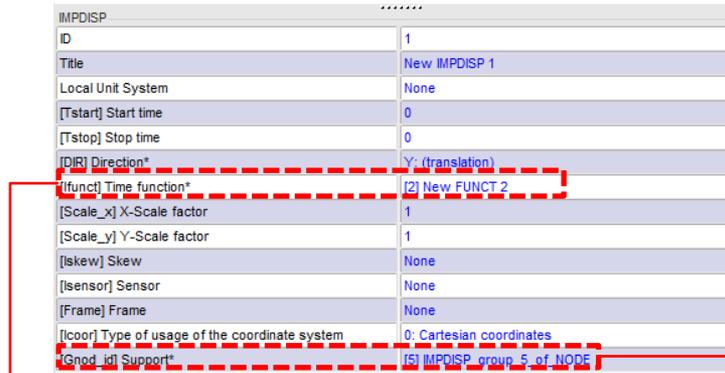
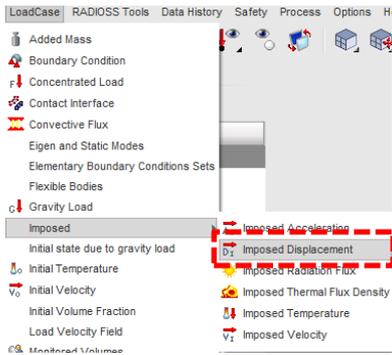
구속조건 설정 (2)

BCS	
ID	2
Title	New BCS 2
ALE formul.	<input type="checkbox"/>
Lagrange mult. formul.	<input type="checkbox"/>
Translation	
[TX] X translation	<input checked="" type="checkbox"/>
[TY] Y translation	<input type="checkbox"/>
[TZ] Z translation	<input checked="" type="checkbox"/>
Rotation	
[RX] X rotation	<input checked="" type="checkbox"/>
[RY] Y rotation	<input checked="" type="checkbox"/>
[RZ] Z rotation	<input checked="" type="checkbox"/>
[Skew_id] Skew	None
[Gnod_id] Support*	1 items



강체 중심 절점은 Y 방향
구속조건을 제외한 모든
자유도 구속
(Tx, Tz, Rx, Ry, Rz 구속)

변위조건 설정



1 LoadCase > Imposed > Imposed Displacement 클릭,
마우스 우클릭 후
Imposed displacements 생성

2 시간-변위 그래프 생성
(0,0), (100,-200)

DIR: Y

3 Support: 앞서 만든 RBE2 요소의 master node 선택

접촉조건 설정



LoadCase RADIOSS Tools Data History

- Added Mass
- Boundary Condition
- Concentrated Load
- Contact Interface

Review

- Create New
 - ALE/Lagrange with sliding (Type 1)
 - Kinematic condition (Type 2)
 - Surface/Surface (Type 3)
 - Nodes/Surface (Type 5)
 - Rigid body/Rigid body (Type 6)
 - Multi usage (Type 7)
- Create Template
- Delete
- Data History
- Clone
- Change Subtype



LoadCase >
Contact Interface 클릭,
마우스 우클릭 후 "Multi
usage (Type 7) 클릭



TYPE7	
ID	1
Title	New INTER 1
Self Impact	<input checked="" type="checkbox"/>
Local Unit System	None
Formulation	0: Classical
[lstf] Stiffness definition	0: Default, set to value defined in /DEFAULT/INTER/TYPE
[the] Heat contact	<input type="checkbox"/>
[lgap] Gap/element option	0: Default, set to value defined in /DEFAULT/INTER/TYPE
[Fpenmax] Maximum fraction of initial penetration	0
[lbag] Vent hole closure when contact	0: Default, set to value defined in /DEFAULT/INTER/TYPE
[idel] Node and segment deletion	0: Default, set to value defined in /DEFAULT/INTER/TYPE
[lcurv] Slave gap with curvature	0: No curvature
[ladm] Local curvature flag	0: Not activated
[Stfac] Scale factor for stiffness	1
[dtmin] Limiting nodal time step	0
[rem_gap] Flag for deactivating slave nodes if element s	0: Default, set to value defined in /DEFAULT/INTER/TYPE
[rem_c2] Flag for deactivating the slave node, if the sam	0: default, set to the value defined in /DEFAULT/INTER/T
[Gapmin] Min. gap for impact activ.	.9
[_BC] Bound. cond. deactivation	
[lnact] Stiffness deactiv. (init. penetration)	0: Default, set to value defined in /DEFAULT/INTER/TYPE
[VisS] Critical damping coeff. on interface stiffness	0
[VisF] Critical damping coeff. on interface friction	0
[Tstart] Start time	0
[Tstop] Stop time	0
[Bumult] Sorting factor	0
[fric] Friction formulation	0: Static Coulomb
[Fric] Coulomb friction	0.2
[iform] Friction penalty formulation	0: Default, set to value defined in /DEFAULT/INTER/TYPE
[filtr] Friction filtering	0: No filtering
[sens_ID] Sensor to Activate/Deactivate the interface	None
[fct_ID] Friction coefficient with temperature function id	None
[AscaleF] Abscissa scale factor on FCT_IDK	0
[fric_ID] Friction identifier for friction definition for select	None
[Mast_id] Master surface*	[8] INTER_group_8_of_PART
[_Mast_id] Master surface (Advanced selector)	

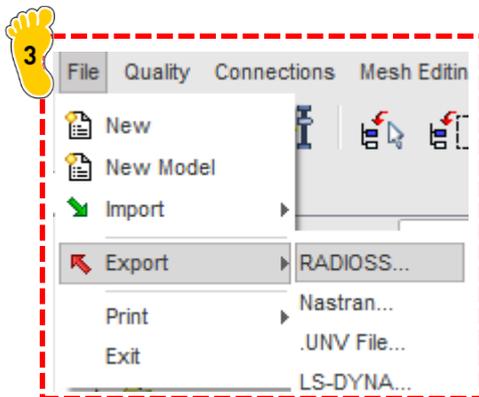
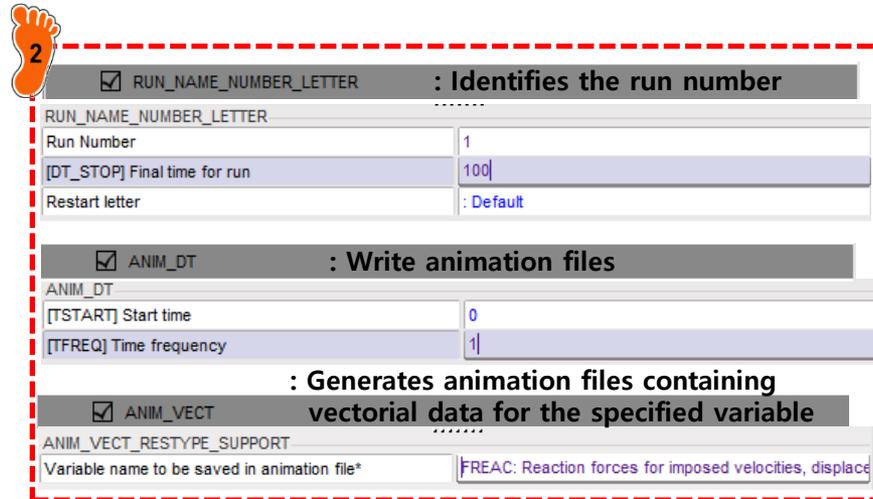
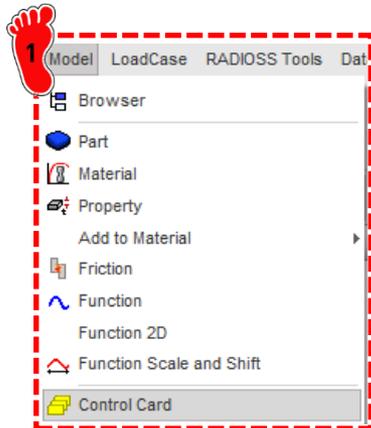


Self Impact 체크
[Stfac] : 1
[Gapmin]: 0.9
[Fric]: 0.2
[Mast_id]: 모델 선택

Interface stiffness scale factor
(1.0 real: contact stiffness is taken from the master segment)

Minimum gap for activation of interface

해석 케이스 설정



1 Model > Control Card 클릭

2 파라미터 설정

3 모델 export (RADIOSS 선택)

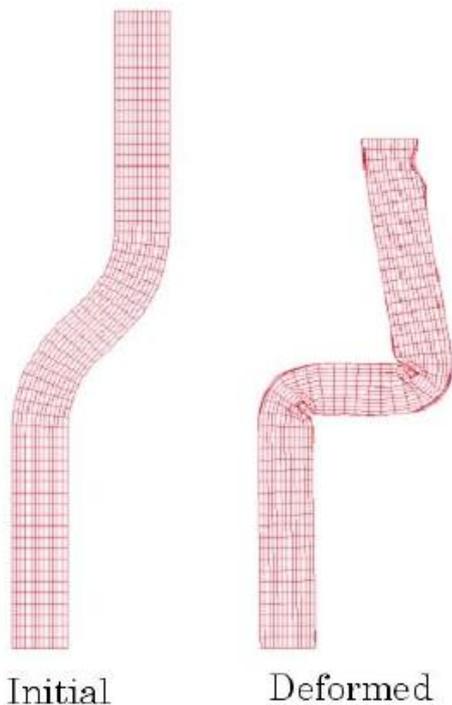
후처리 (1)



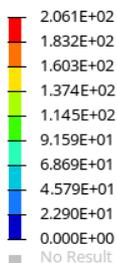
변형된 형상 확인

참고 논문 결과

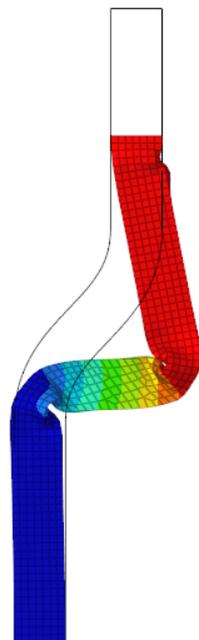
해석 결과



Contour Plot
Displacement(Mag)
Analysis system

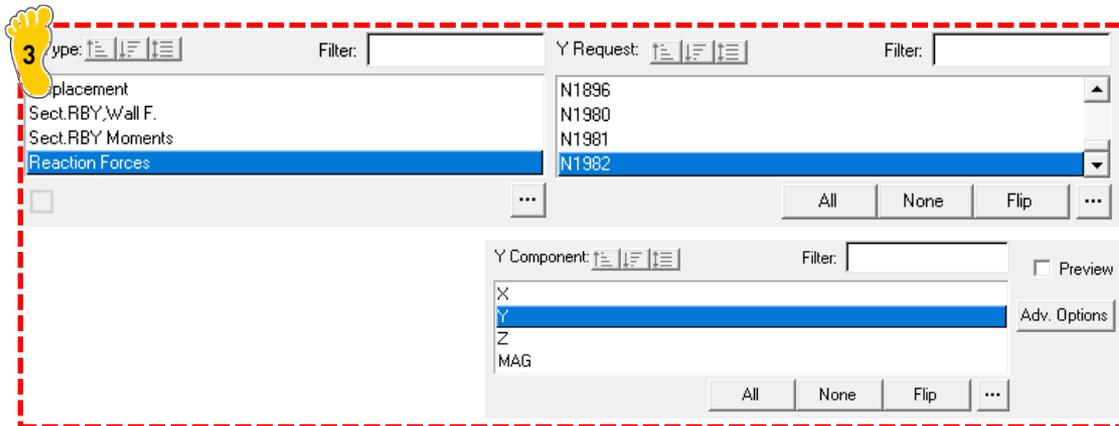
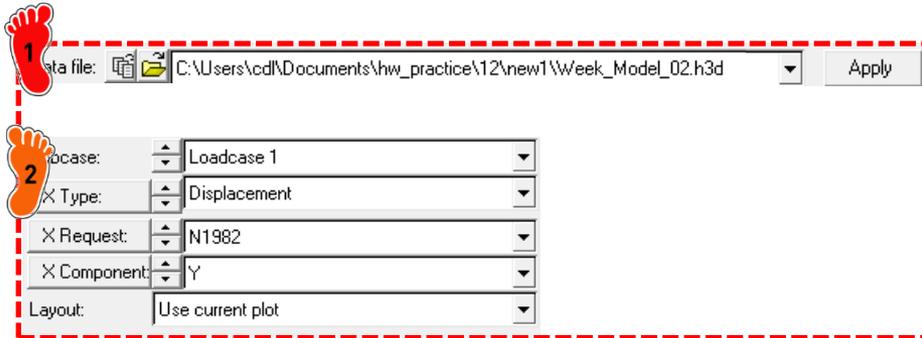


Max = 2.061E+02
Node 1079
Min = 0.000E+00
Node 329



Computational time: 87.70 s

후처리 (2)



Name	Value	Name	Value
Data		Data	
Source	File	Source	File
File	C:\Users\cd\Doc	File	C:\Users\cd\Doc
Subcase	Loadcase 1	Subcase	Loadcase 1
Type	Displacement	Type	Reaction Forces
Request	N1982	Request	N1982
Component	Y	Component	Y
Standard		Standard	
Scale	-1	Scale	-1
Offset	0.000000	Offset	0.000000
First Value	0.000000	First Value	66.874146
Axis Assignment	X1	Axis Assignment	Y1

1 HyperGraph 2D > open H3D 파일

2 X축 > Displacement, N1982, Y 설정

3 Y축 > Reaction Forces, N1982, Y 설정

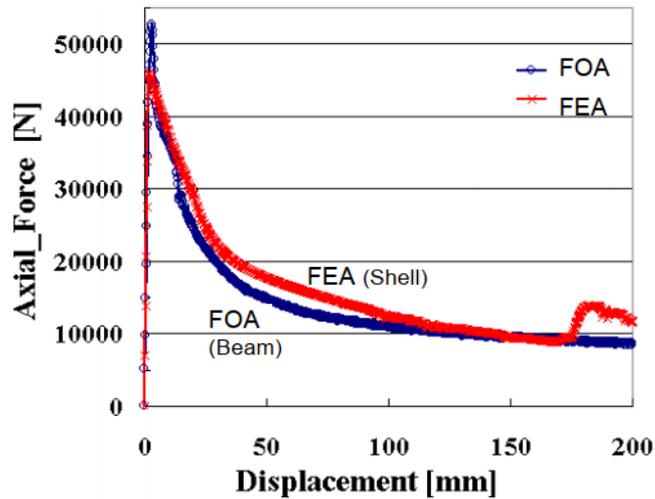
4 X, Y축 Scale > -1 설정 (양수로 전환)

후처리 (2)



강체 중앙 절점(N1982)의
변위 vs. 반력(Y 방향)
그래프 표시

참고 논문 결과



해석 결과

