Explicit Dynamic Analysis (2)

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

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

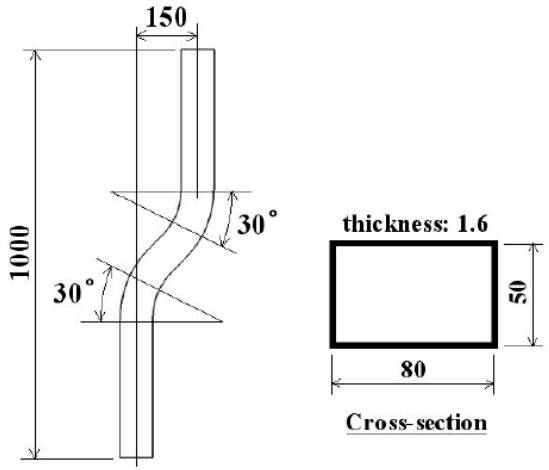
2004-01-1660

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

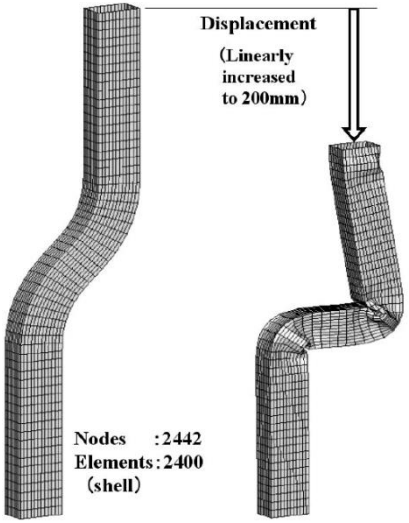
Hidekazu Nishigaki
Toyota Central R&D Labs., Inc.

Noboru Kikuchi
The University of Michigan

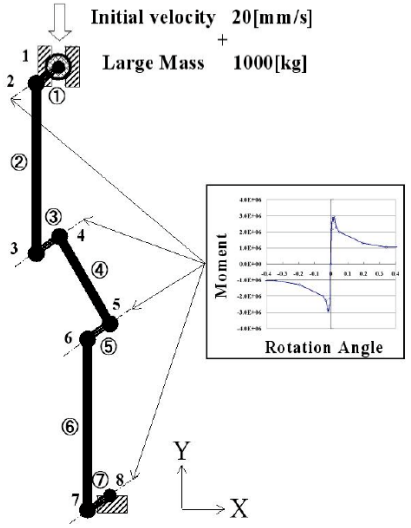
기하 형상 정보



쉘 모델 해석



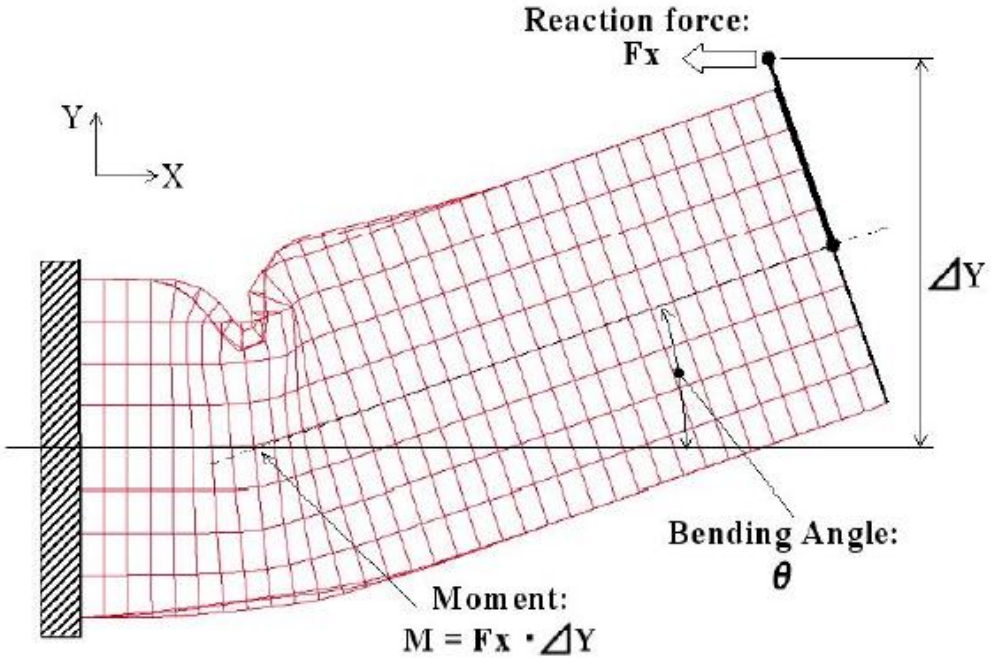
빔 모델 해석

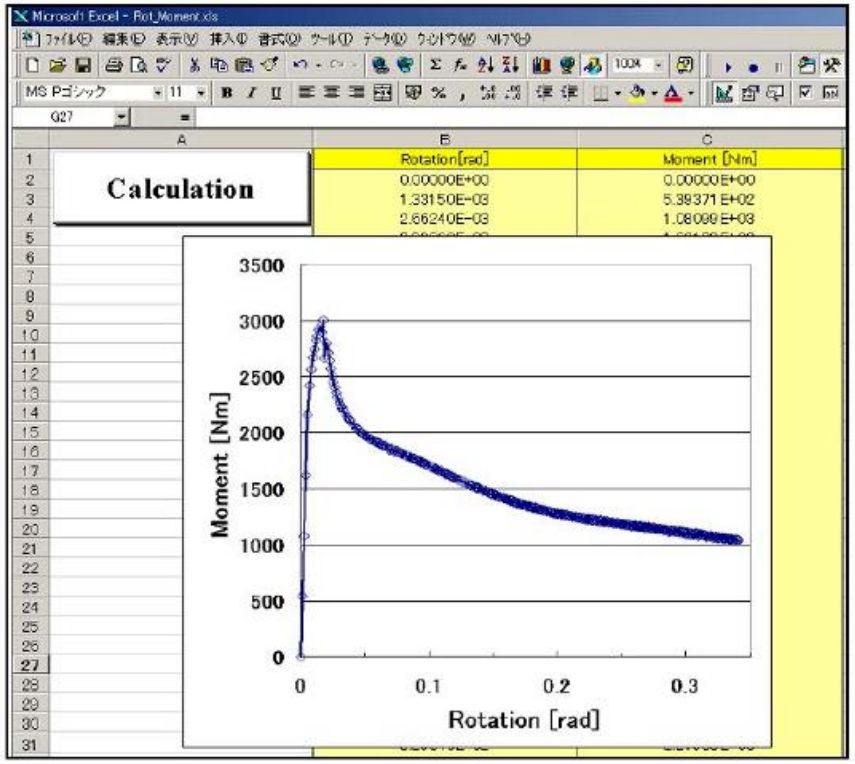


JOINT MOMENT ANALYSIS

예제: JOINT MOMENT ANALYSIS

- Shell 모델의 등가 강성 계산





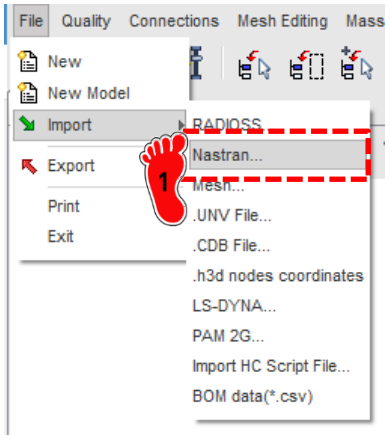
HYPERCRASH 실행

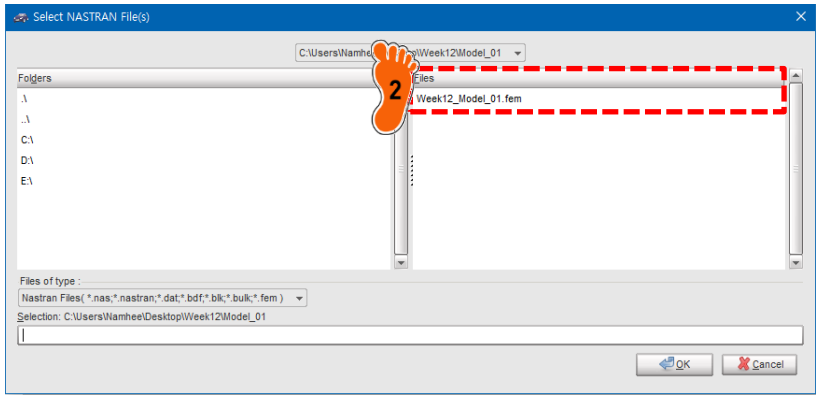
1. HyperCrash2017 실행 후 Working directory 설정 (모델 파일이 있는 폴더로 지정)

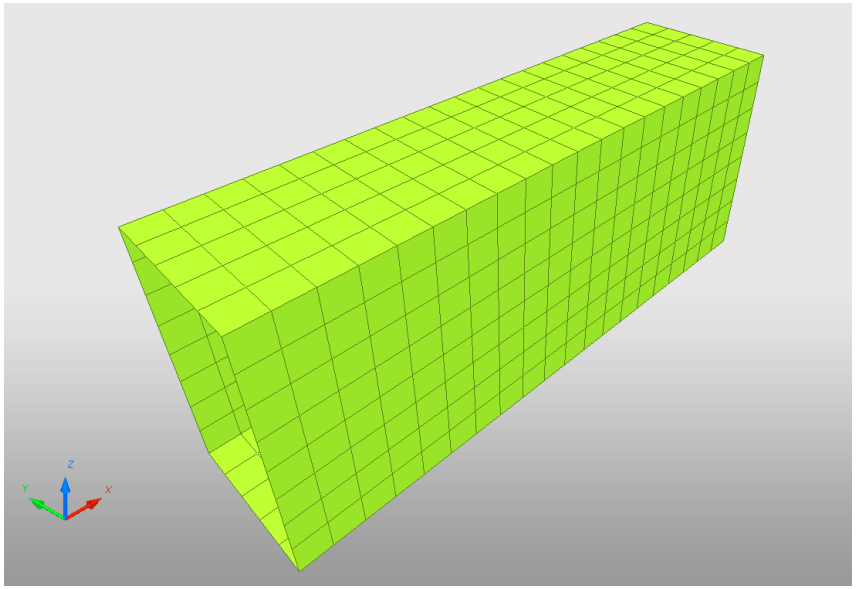
2. User profile, Unit system 설정

3. Run 클릭

유한요소해석 모델 불러오기



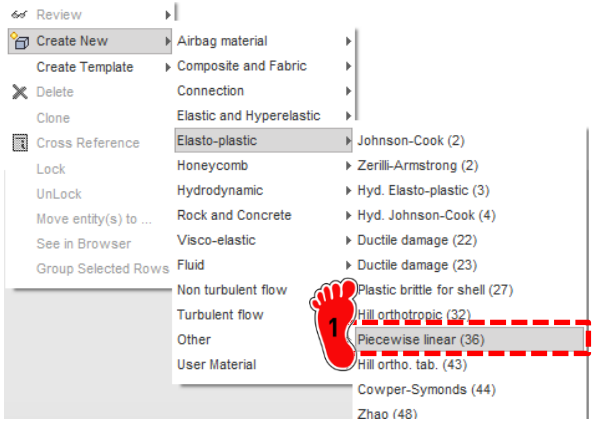




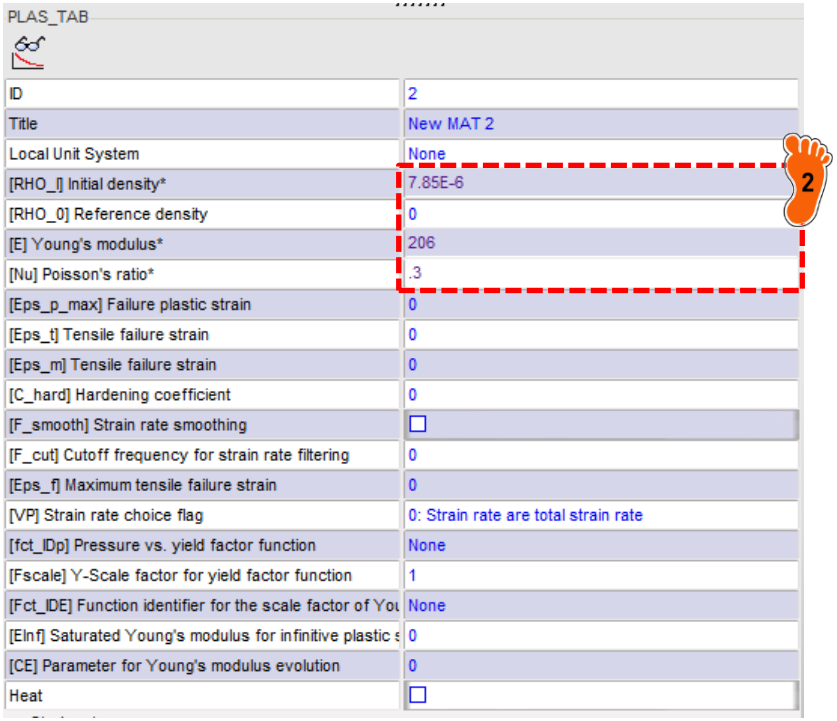
1 File > Import > Nastran 클릭

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)

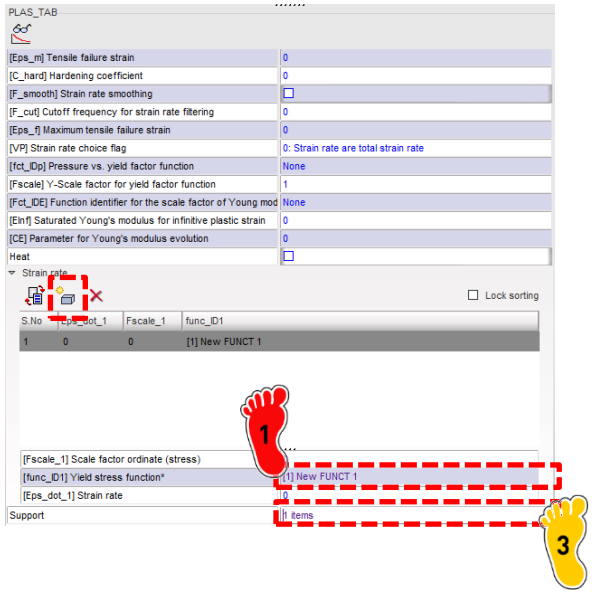


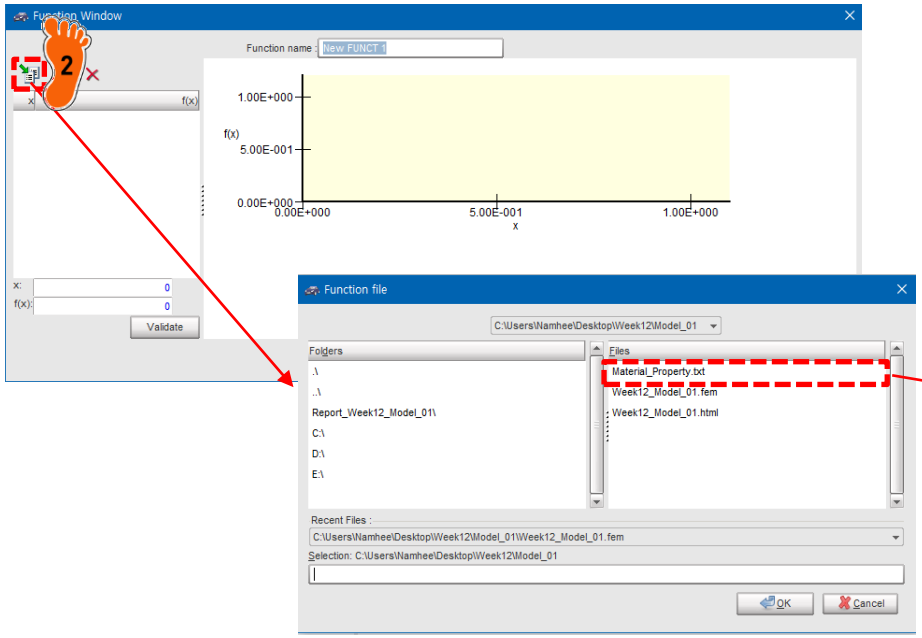
1 Model > Material 클릭 > 마우스 우클릭 후 Create New > Piecewise linear(36) 재료 생성

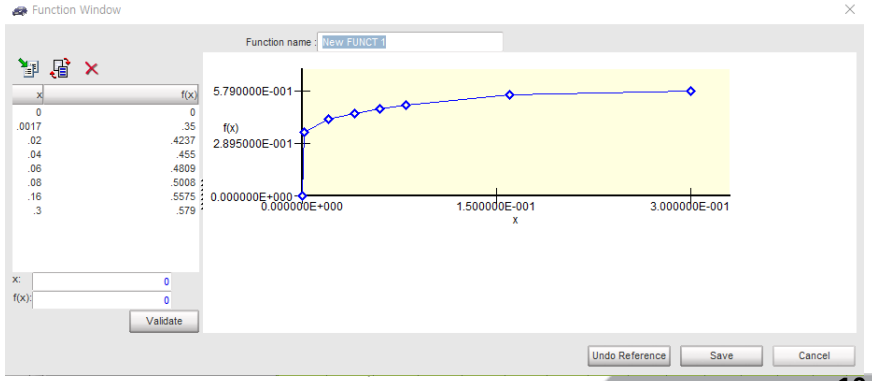
2

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

재료 물성 입력 (2)

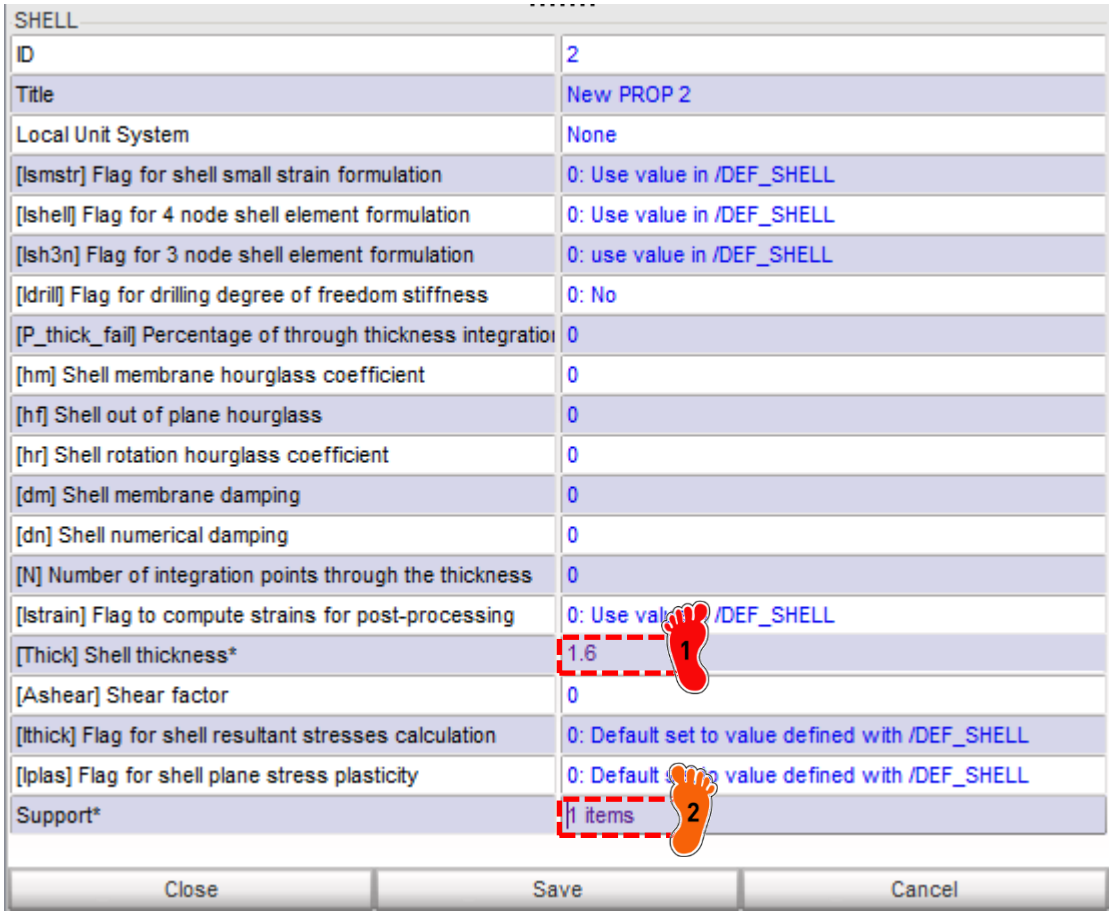






1. Stress-Strain curve 입력 (마우스 우클릭 후 "Define function" 클릭)
2. "Import curve" 클릭 후 "Material Property.txt" 파일 불러오기
3. 모델에 재료 적용

특성 입력



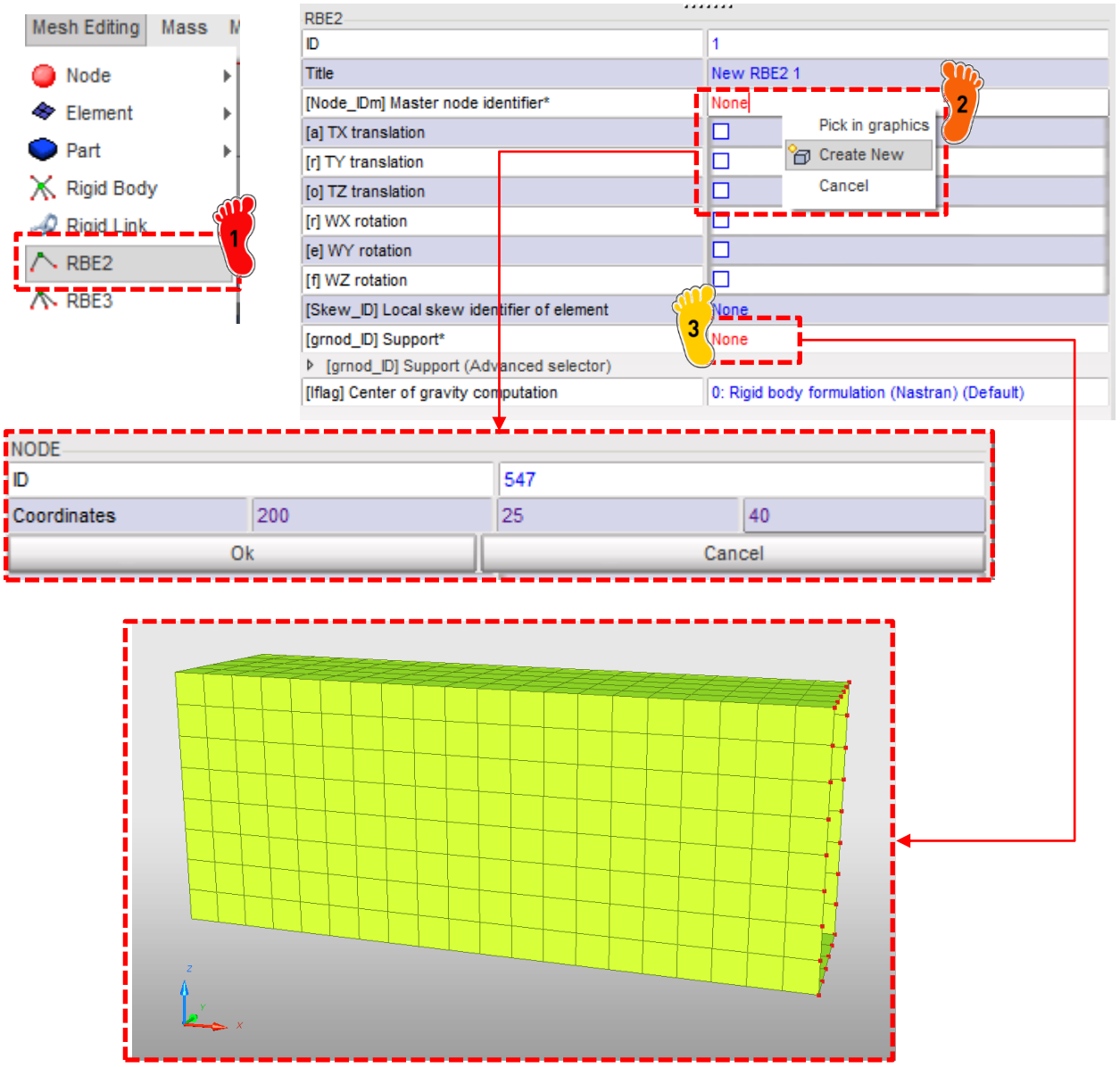
1. Model > Property 클릭, 마우스 우클릭 후 Create New > Surface > Shell (1) 클릭 >

   두께 1.6mm 입력

2. 모델에 특성 적용

Help:
/PROP/TYPE1 (SHELL)

RBE2 생성 (1)

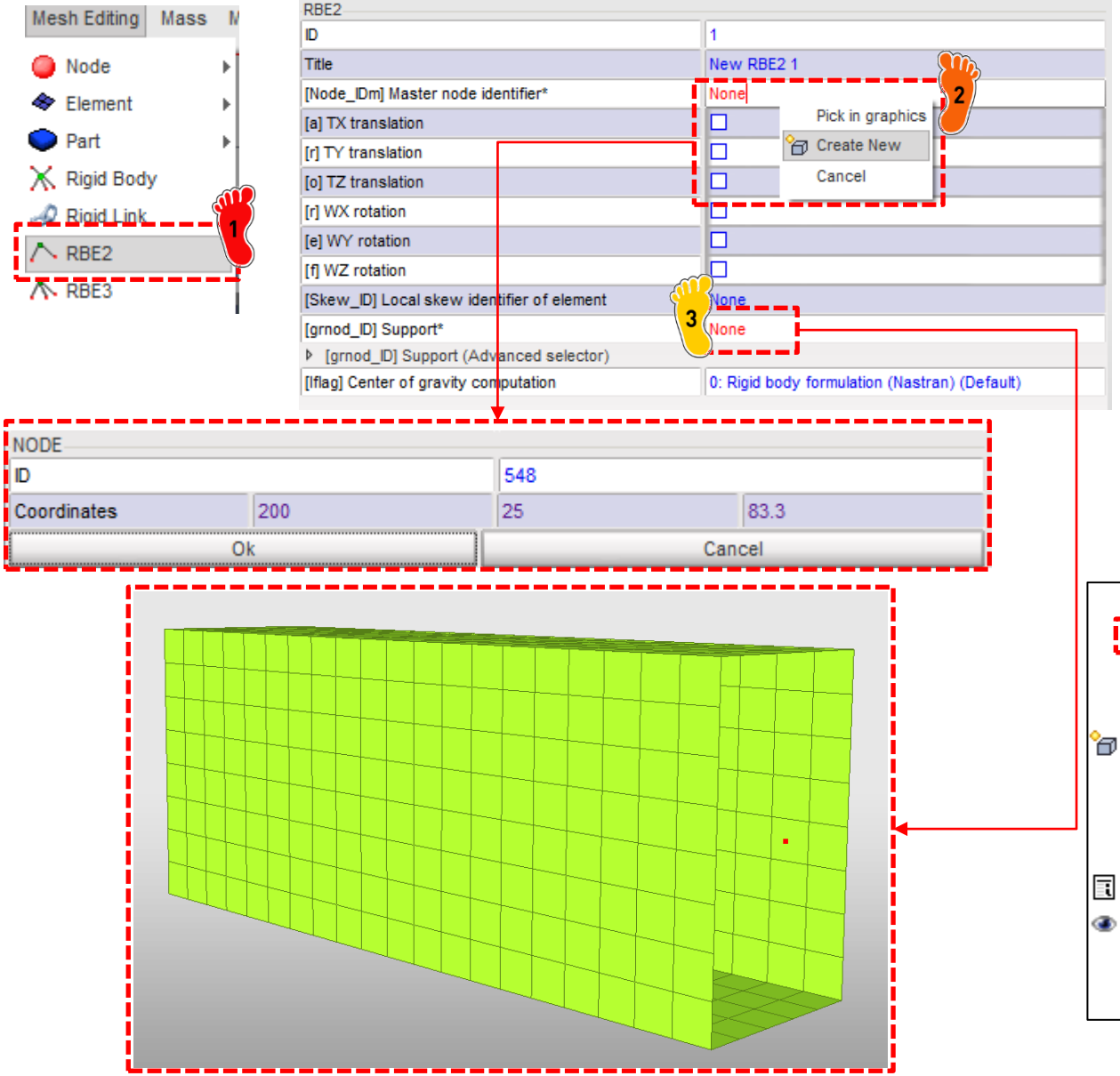


1 Mesh Editing > RBE2 생성 > Create New

2 Master node 좌표: (200, 25, 40)

3 Support: X방향 끝단 노드 선택

RBE2 생성 (2)



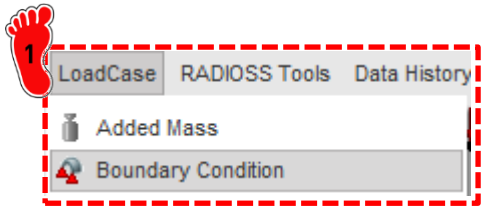
1 새로운 RBE2 생성

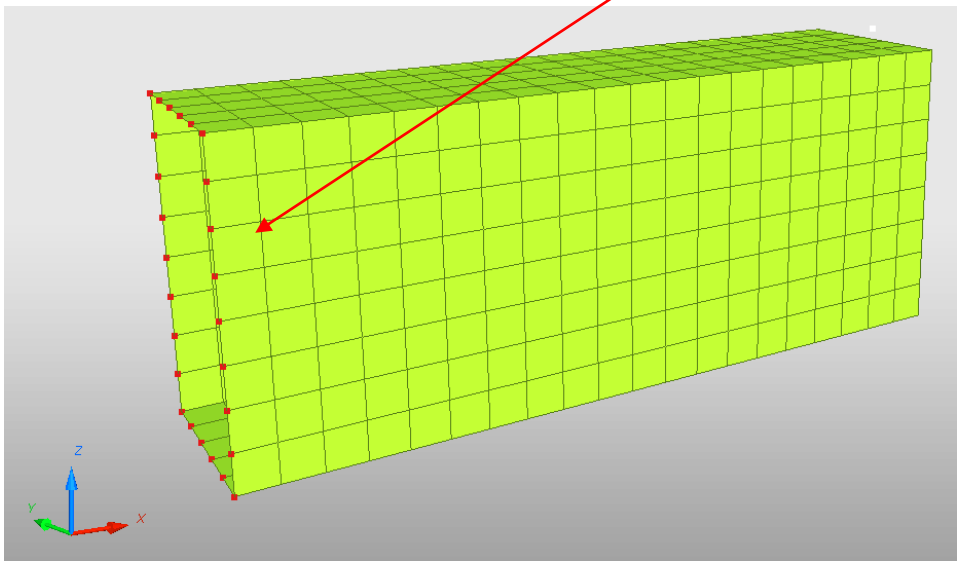
2 Master node 좌표: (200, 25, 83.3)

3 Support: 이전에 만든 RBE2 mater node (no.547)

(F8 : free node display on/off)

구속조건 설정



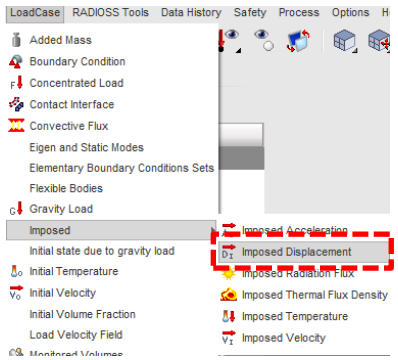


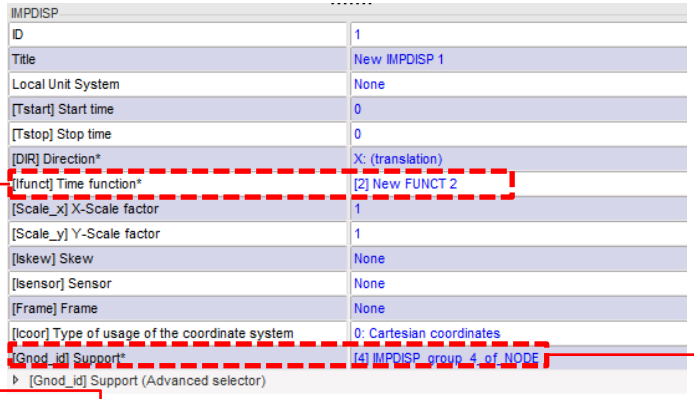
1 LoadCase > Boundary Condition 클릭, 마우스 우클릭 후 Create new 클릭

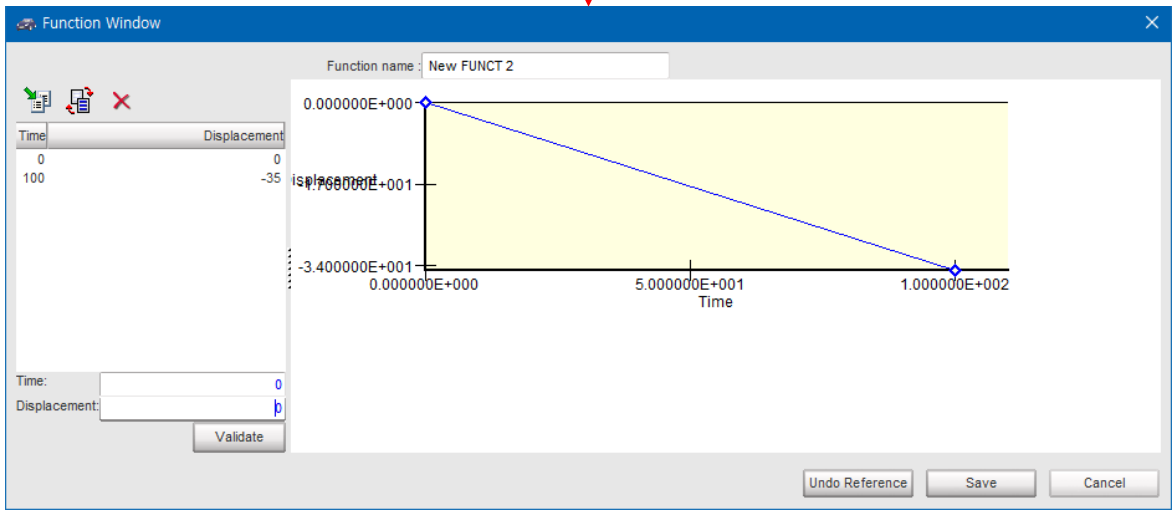
2 완전 구속 설정

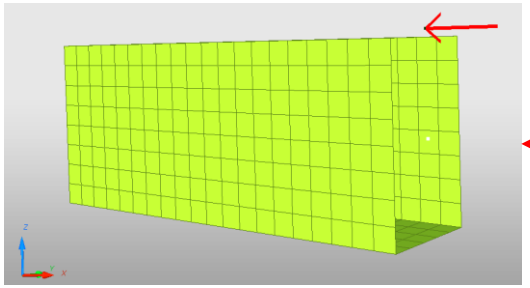
3 구속할 절점 선택

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

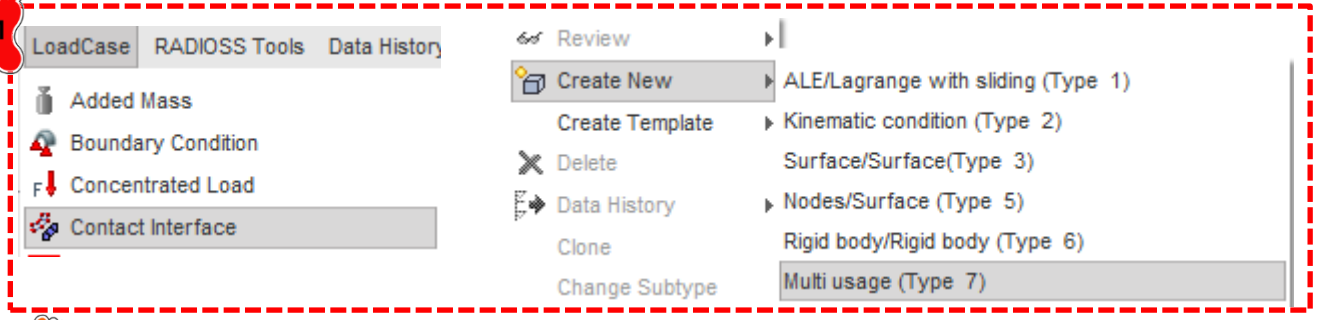
접촉조건 설정

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



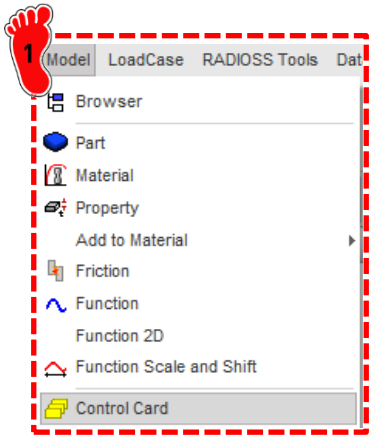
1 LoadCase > Contact Interface클릭,
마우스 우클릭 후 "Multi usage (Type 7) 클릭

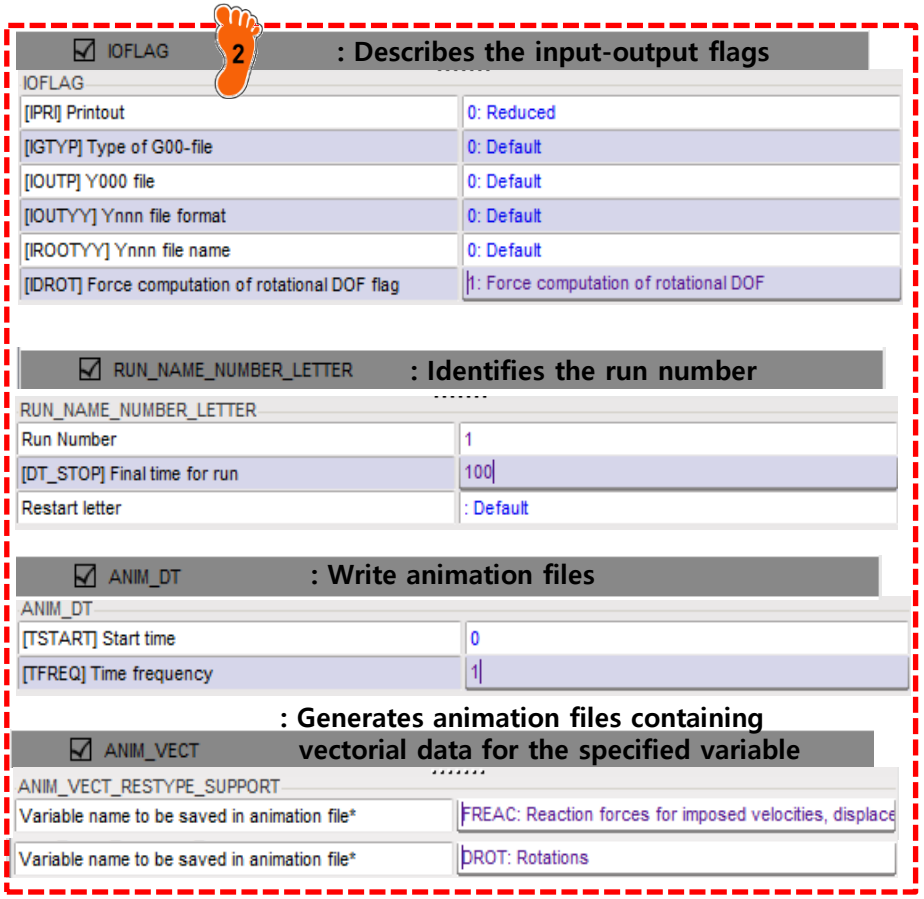
2 Self Impact 체크
[Stfac] : 1
[Gapmin]: 0.9
[Fric]: 0.2
[Mast_id]: 모델 선택

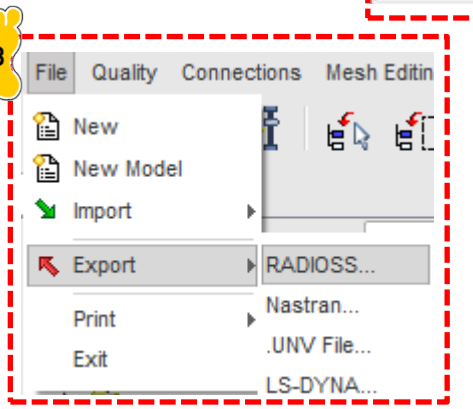
Interface stiffness scale factor
(1.0 real: contact stiffness is taken from the master segment)

Minimum gap for activation of interface

해석 케이스 설정







1 Model > Control Card클릭

2 파라미터 설정

3 모델 export (RADIOSS 선택)

해석 실행 (RADIOSS)

1 RADIOSS 2017

2 HyperWorks Solver Run Manager (@DESKTOP-L2LNH62)

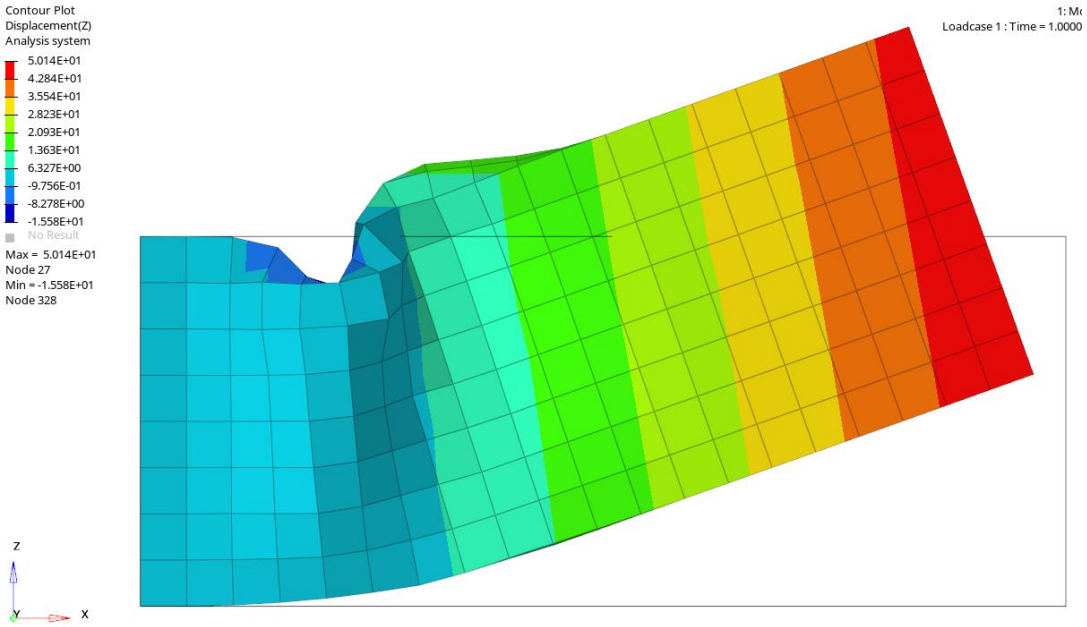
File Edit View Logs Solver HyperWorks Help

Input file(s): Week12_Model_01_0000.rad

Options:

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

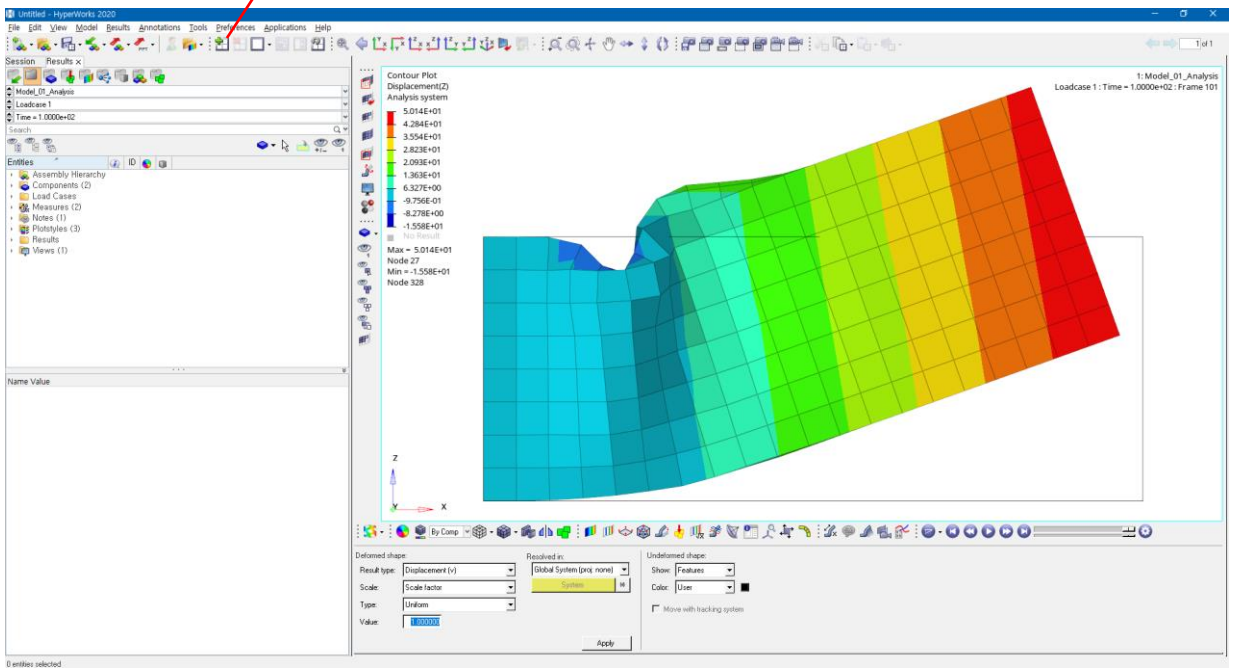
Run Close



1 Radioss2020 실행

2 앞서 제작한 모델 불러온 뒤 Run 클릭

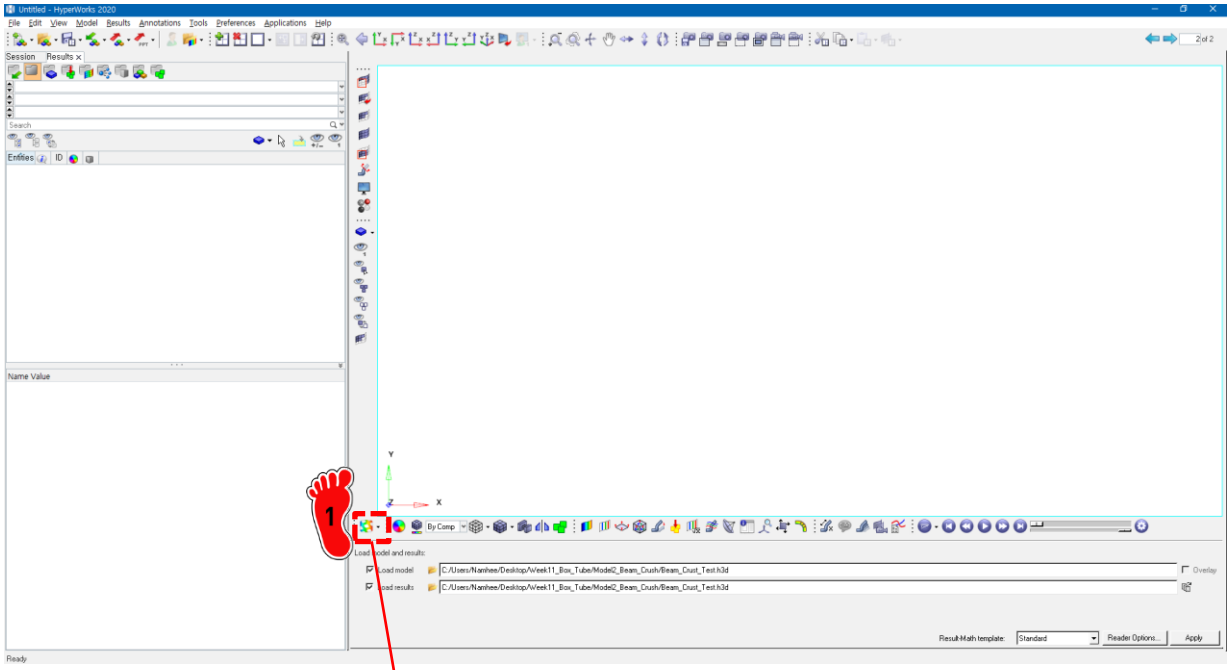
후처리 (1)

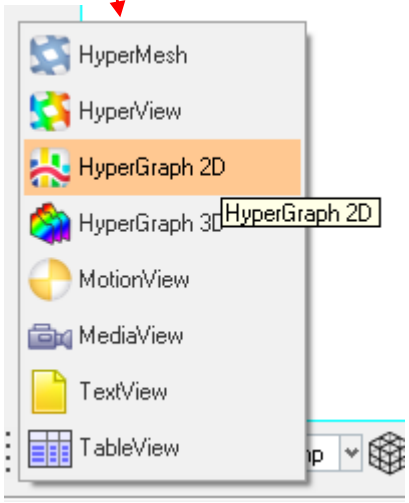


1 HyperView에서 Add Page 버튼 클릭

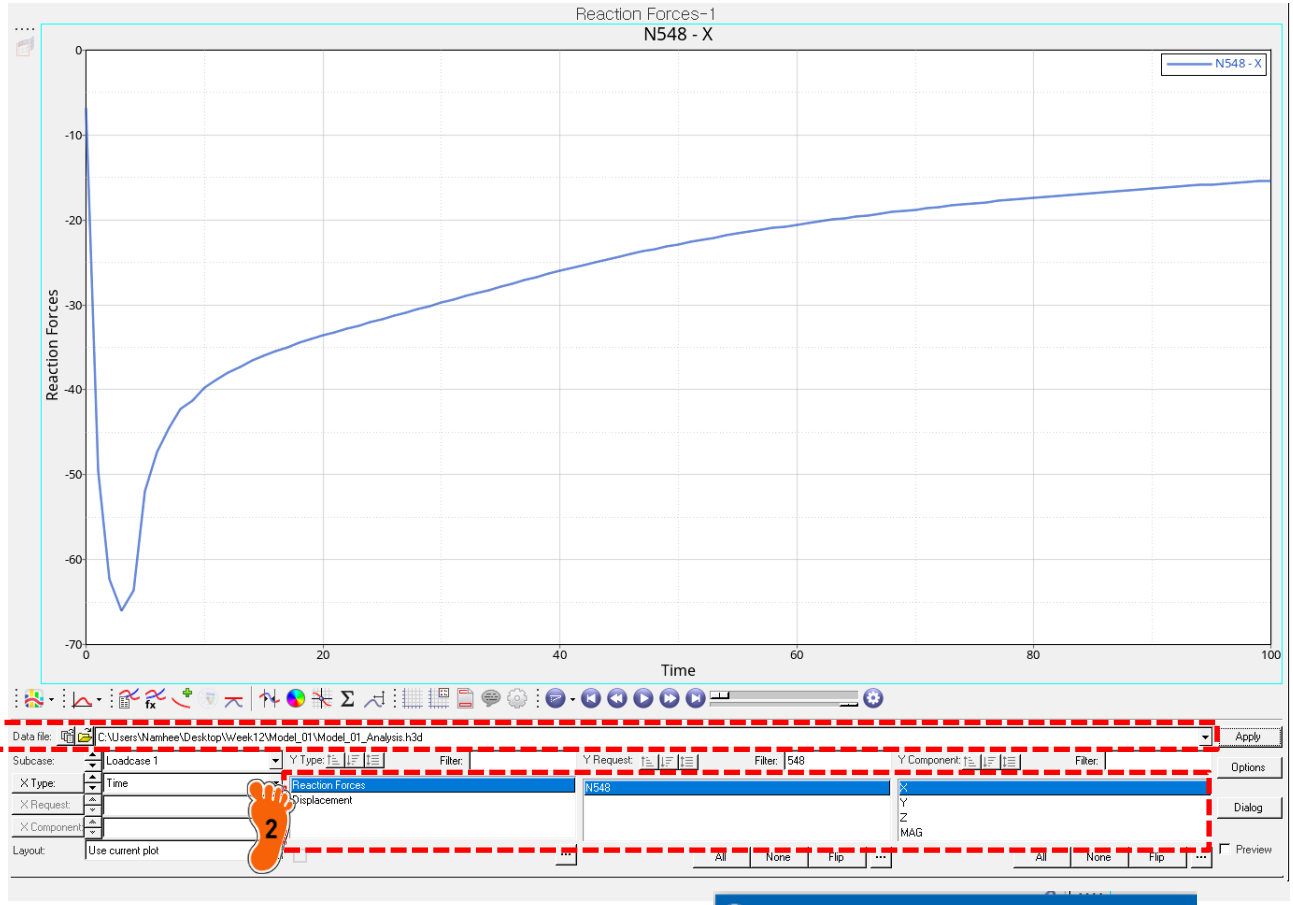
후처리 (2)

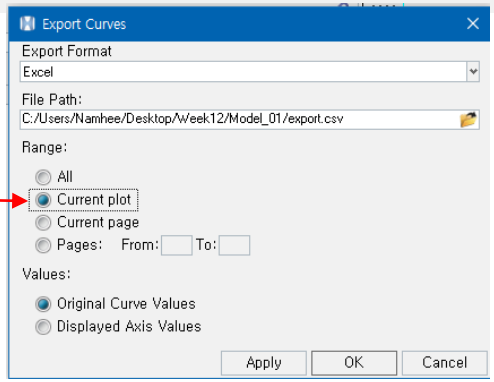
1 HyperGraph 2D 클릭





후처리 (3)



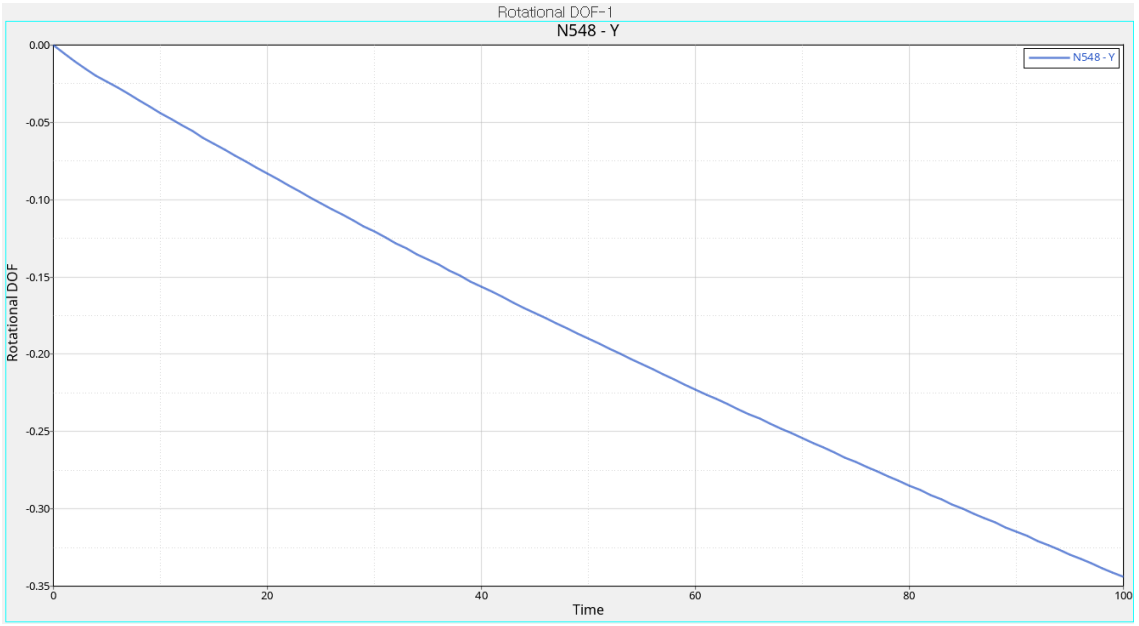


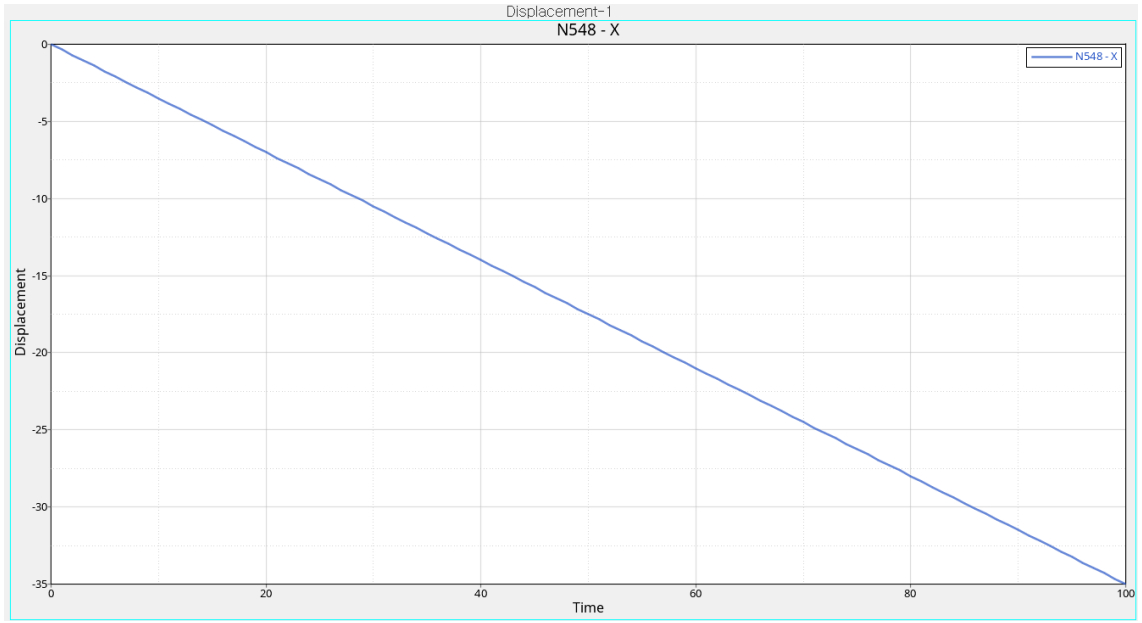
1 해석 결과 파일 불러오기 (*.h3d 파일)

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

3 그래프 csv파일로 저장

후처리 (4)



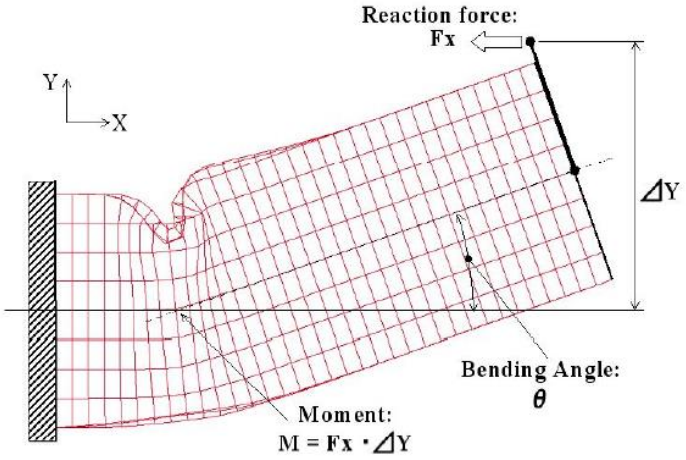


1. 새로운 그래프 추가 후
Rotational DOF 선택,
548번 절점 선택,
Y 방향으로의 그래프 확인
→ 그래프 csv파일로 저장

2. 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
1	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

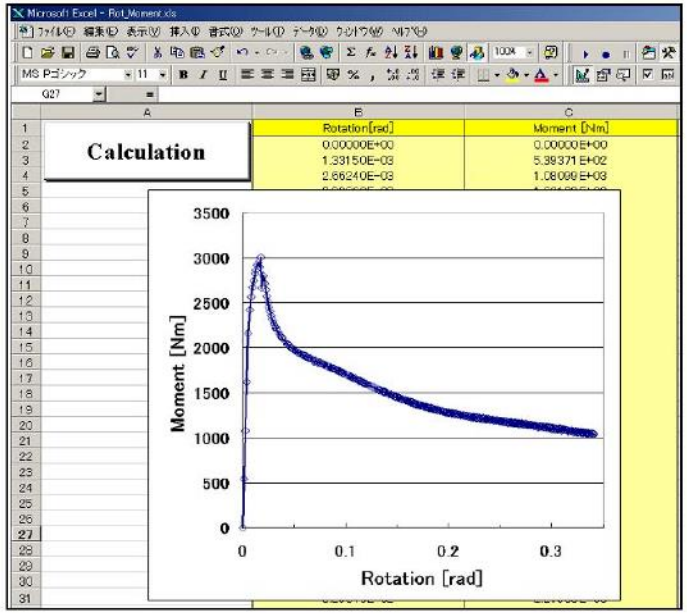


∇Y = (displacement X)*(rotation)
Moment = (reaction force)*(del Y + 43.3)

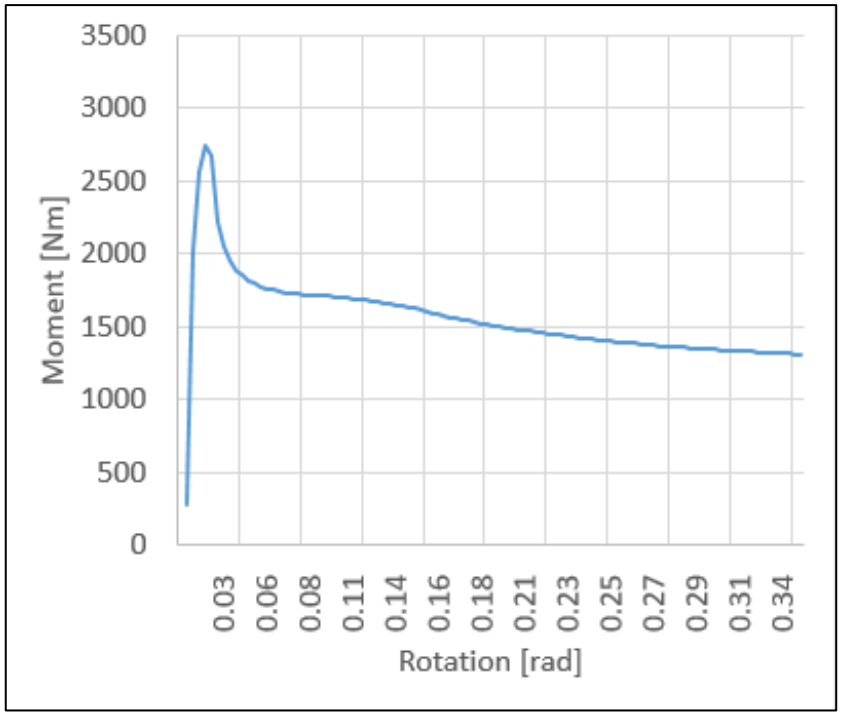
1 Excel을 통해 Moment 계산

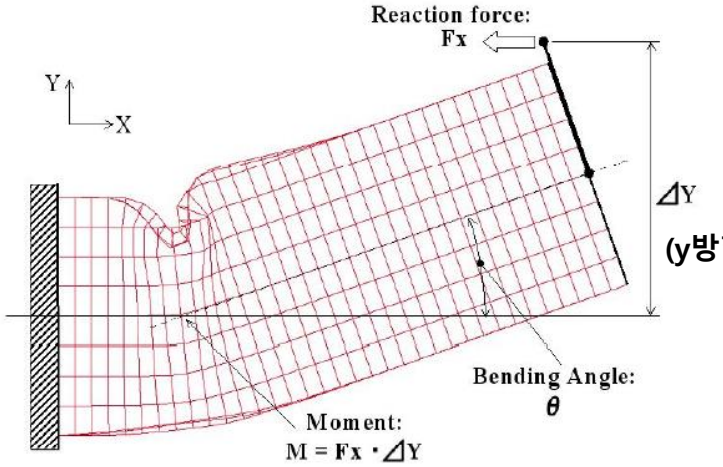
후처리 (5)

참고 논문 결과



해석 결과

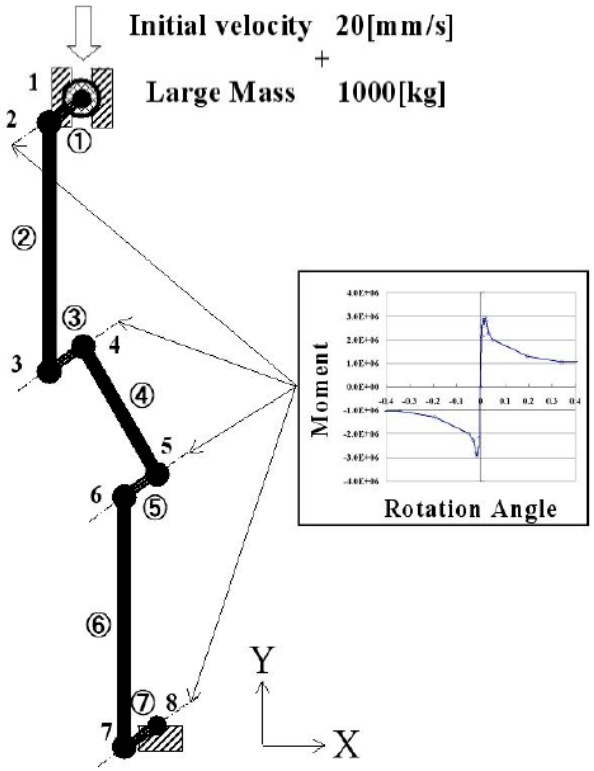


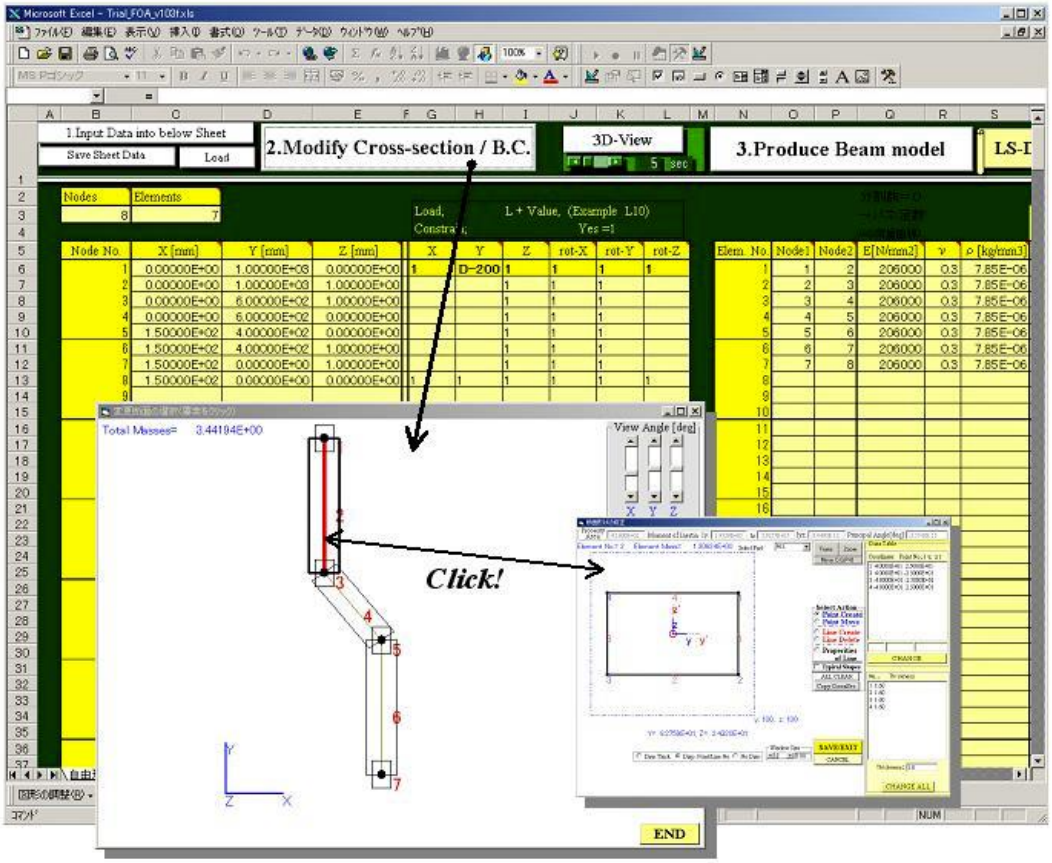


(y방향 변위 + 43.3)

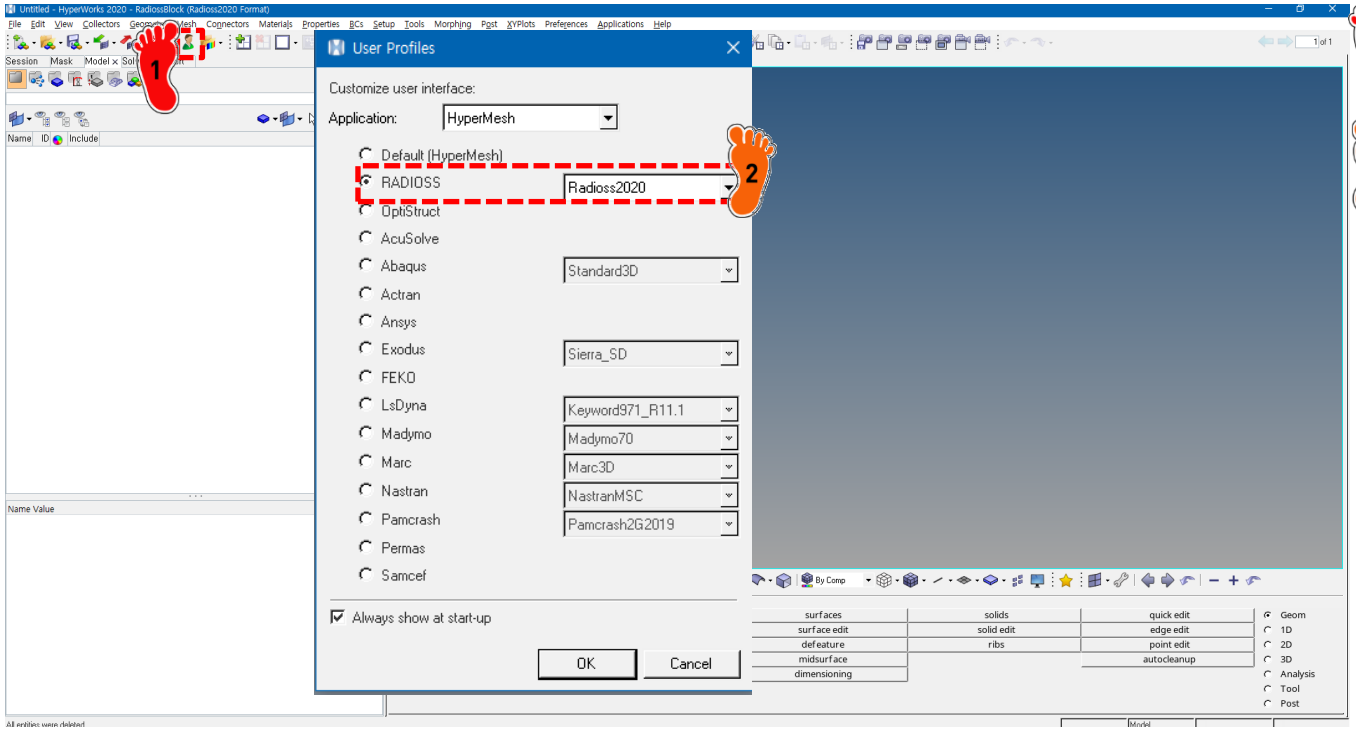
MID RAIL ANALYSIS 1 (BEAM ELEMENT)

예제: JOINT MOMENT ANALYSIS





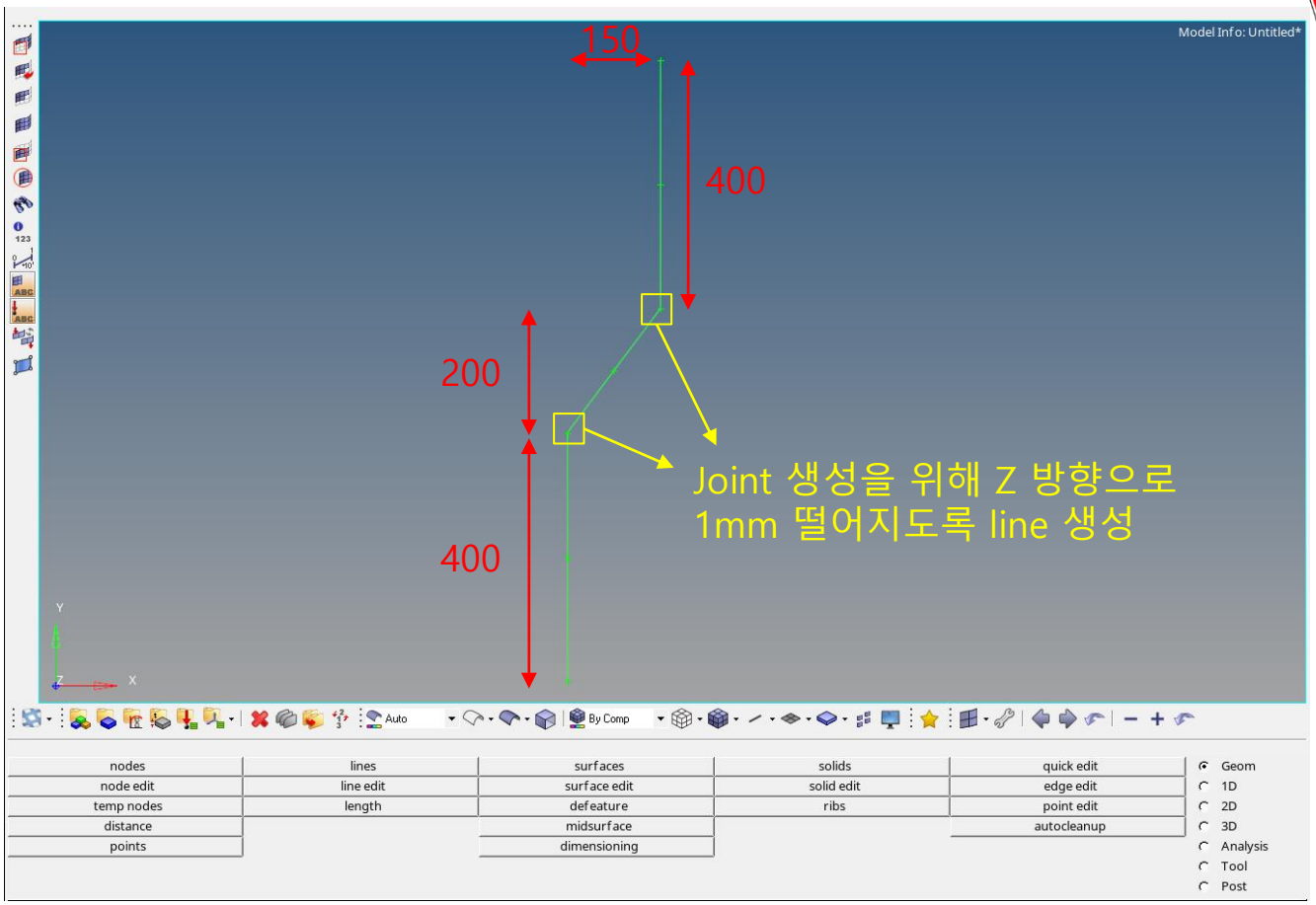
기하형상 생성 (1)



1 User Profiles ( ) 클릭

2 RADIOSS 선택

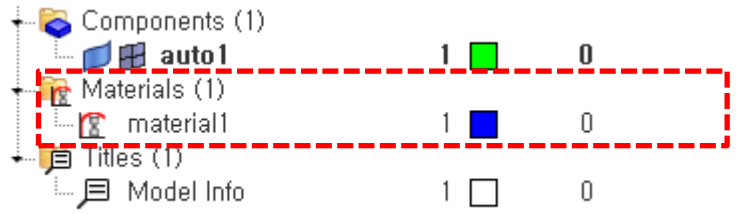
기하형상 생성 (2)



1 3개의 line 생성

(0,0,0) – (0,400,0)
(0,400,1) – (150,600,1)
(150,600,0) – (150, 1000, 0)

재료 입력



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

1 선형 재료 생성
Card Image: M1_ELAST
Rho_Initial: 7.85e-9
E: 206 GPa
Nu: 0.3

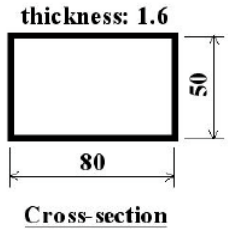
요소망 생성 (1)

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

Name	Value
Solver Keyword	/PROP/BEAM/
ID	1
Name	property1
Color	
Include	[Master Model]
Defined	☑
User Comments	Hide In Menu/Export
Card Image	P3_BEAM
Hyperbeamsection	(1) beamsection1
Regular_OR_encrypted_flag	Regular

Name	Value
Solver Keyword	/PART/
ID	1
Name	auto1
Color	
Include	[Master Model]
Defined	☑
User Comments	Hide In Menu/Export
Card Image	Part
Prop_Id	(1) property1
Mat_Id	(1) material1

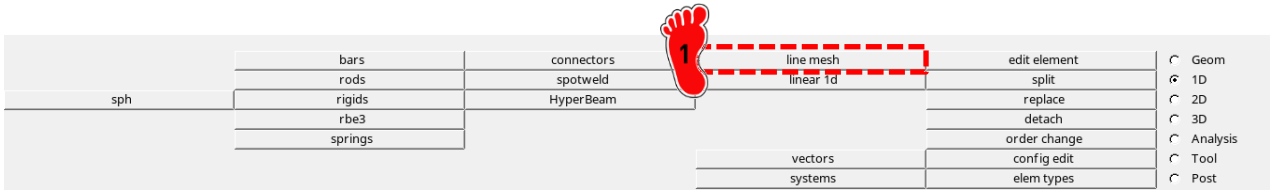
1 Beam section 생성

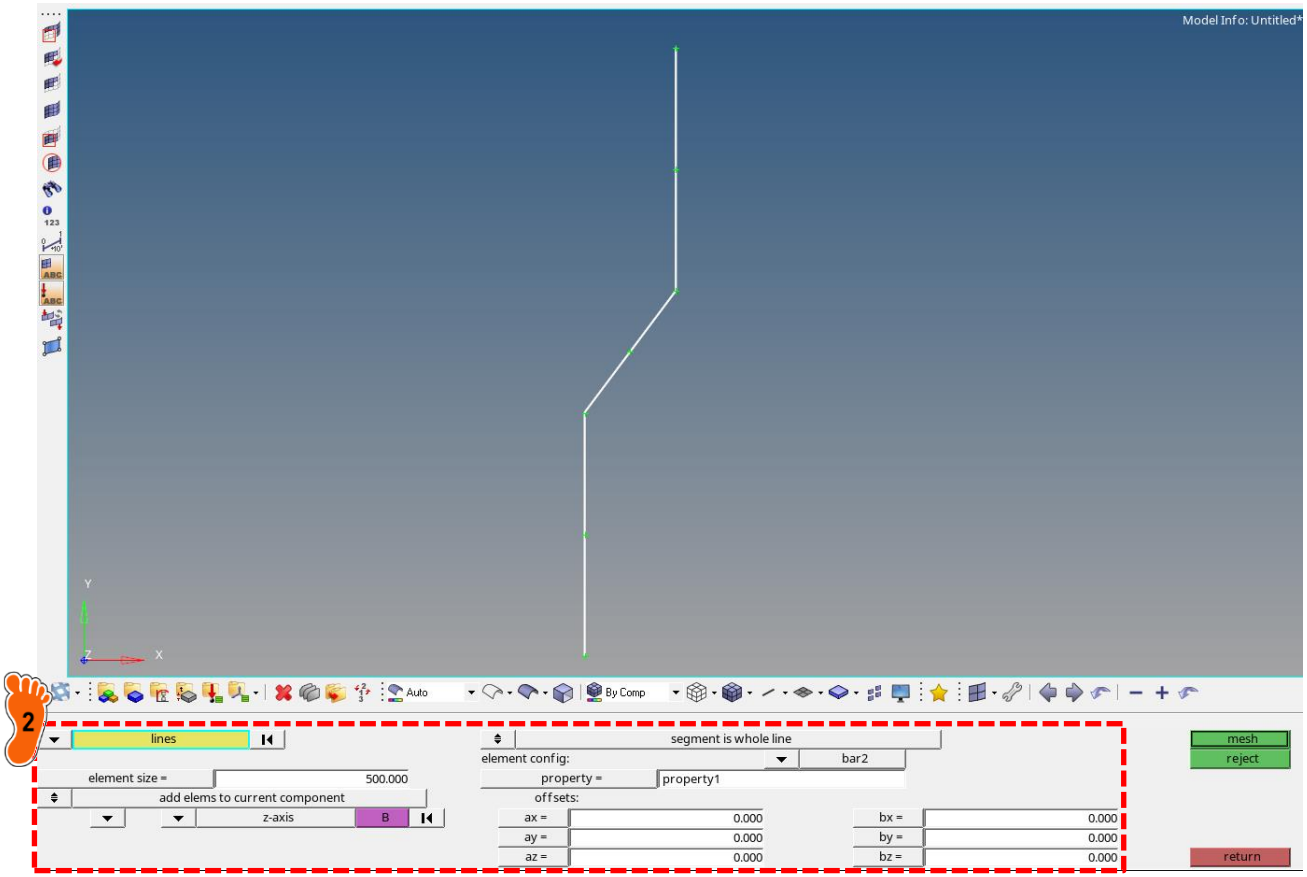


2 Property 생성
Card Image: P3_BEAM
Beam section 설정

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	☑
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

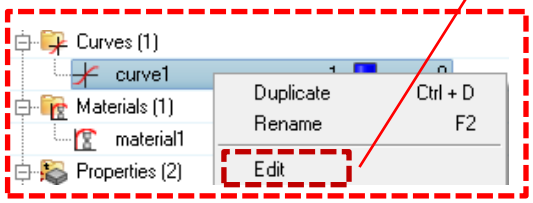
Tx,Ty,Tz,Rx,Ry:
K_TensX 등 = 1e10 (K1~5)

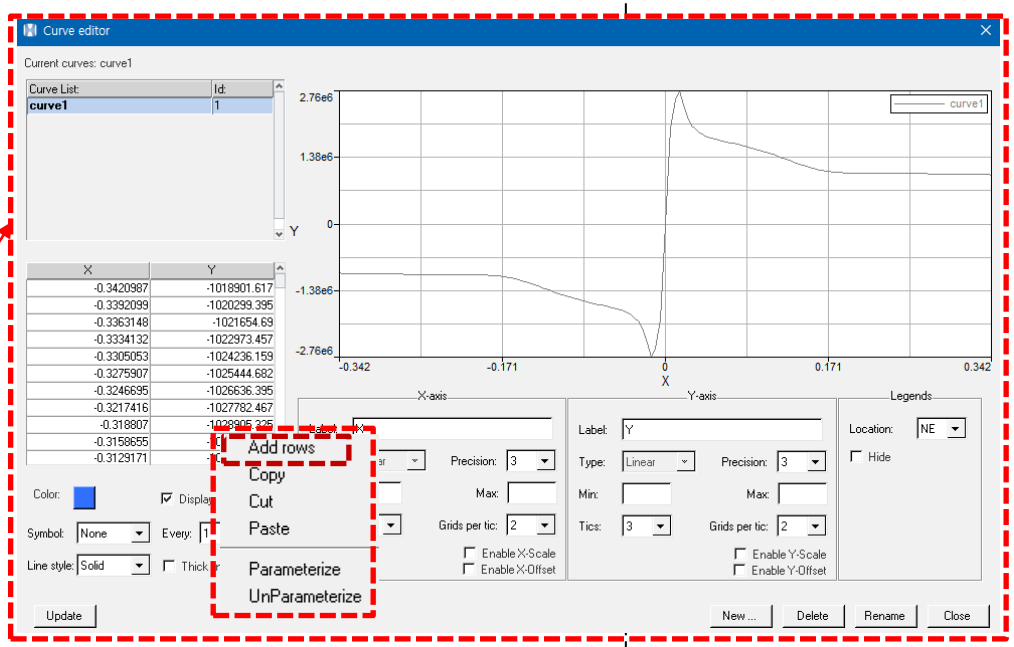
2

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	
Israte	

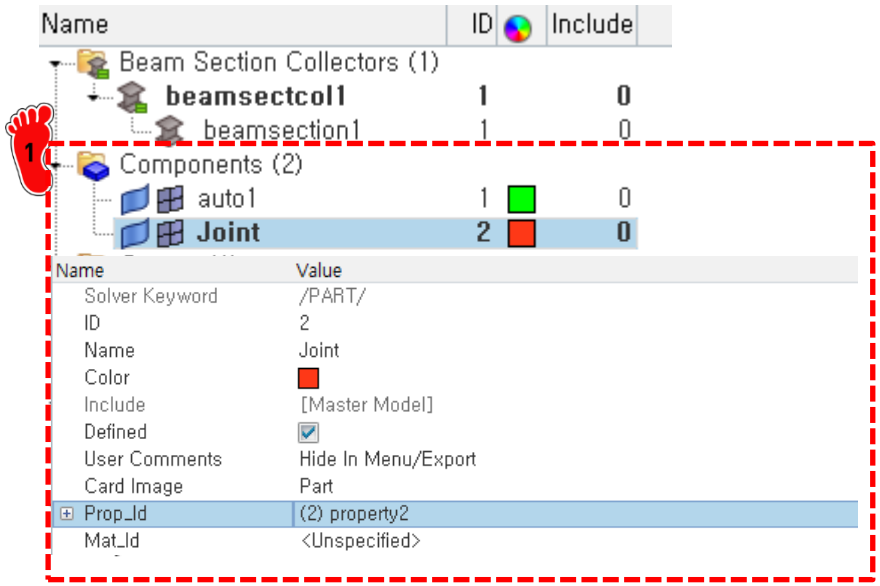
Create
Edit
Plot Curve
Show
Hide
Isolate Only
XRef entities
✔ Filter entities
✔ Warn upon entity type change

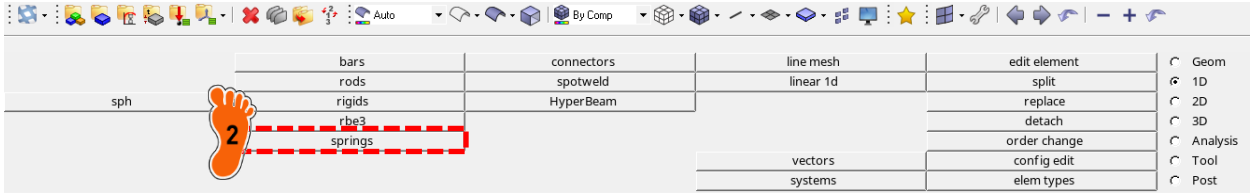
2 Rz: K_TorsZ = 0 (K6),
fct_IDi7(rotation in Z)
활성화하여 회전 강성 테이블 입력

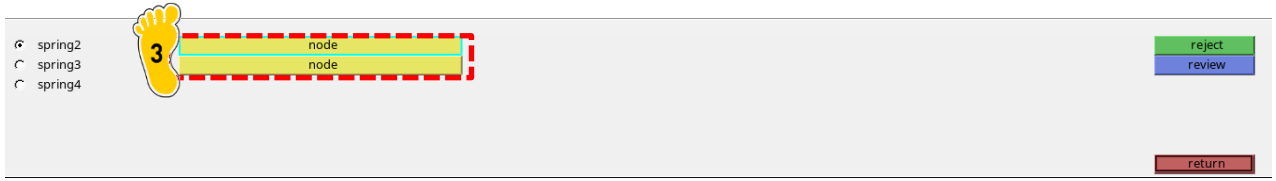




조인트 요소 생성 (2)

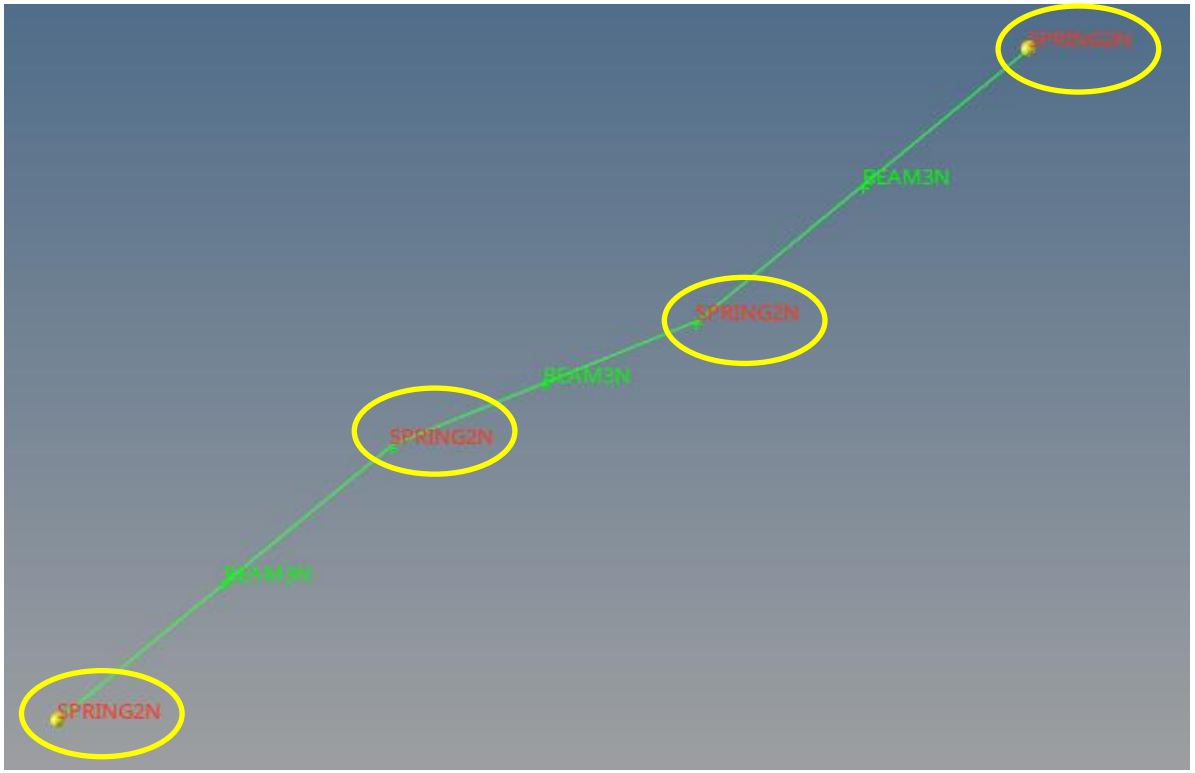






1. Joint component 생성 후 property 부여
2. 1D>spring
3. spring2 생성에서 절점 두 개 선택하여 joint 요소 생성

조인트 요소 생성 (3)

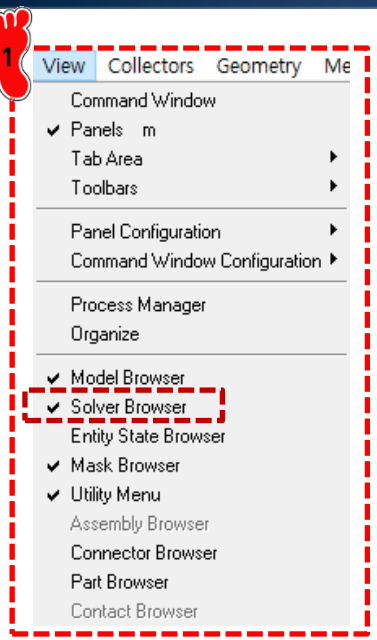


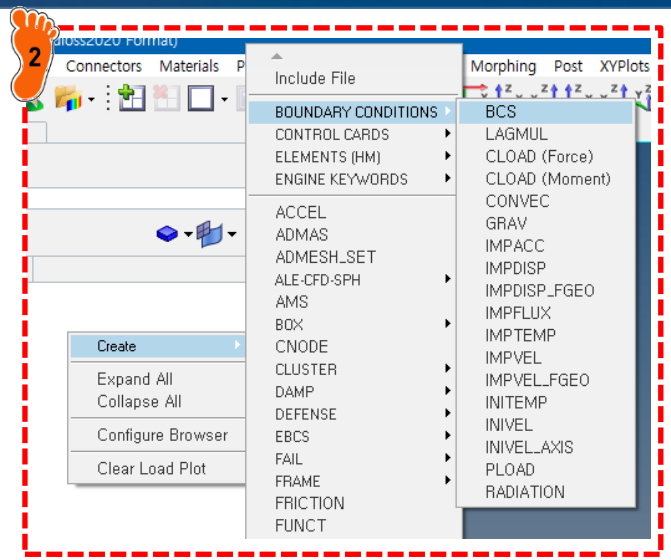
1 네 군데 joint 요소 생성

2 양 끝단에는 temp node를 생성하여 joint를 생성함

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

구속조건 설정 (1)



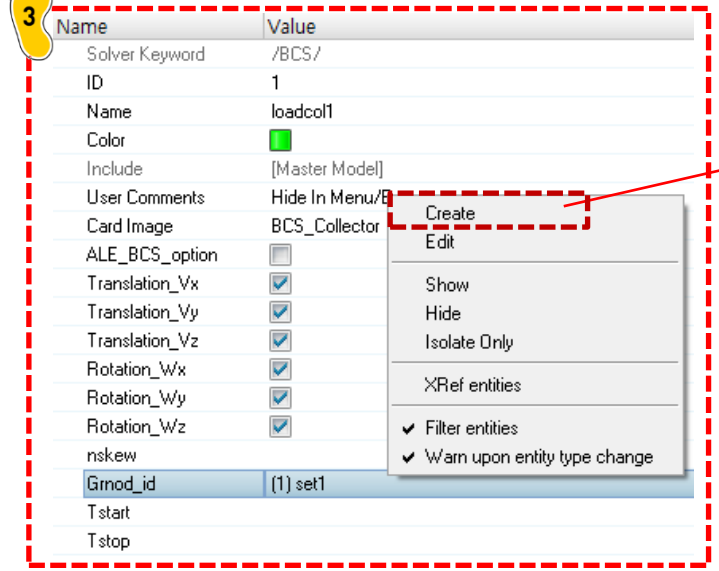


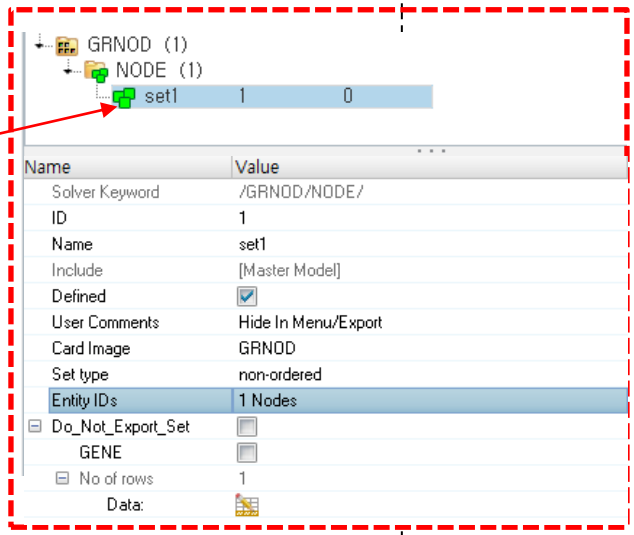
1 View>Solver Browser 체크

2 Solver 탭에서 마우스 우클릭 하여 BCS 생성

DOF 1~6 모두 구속

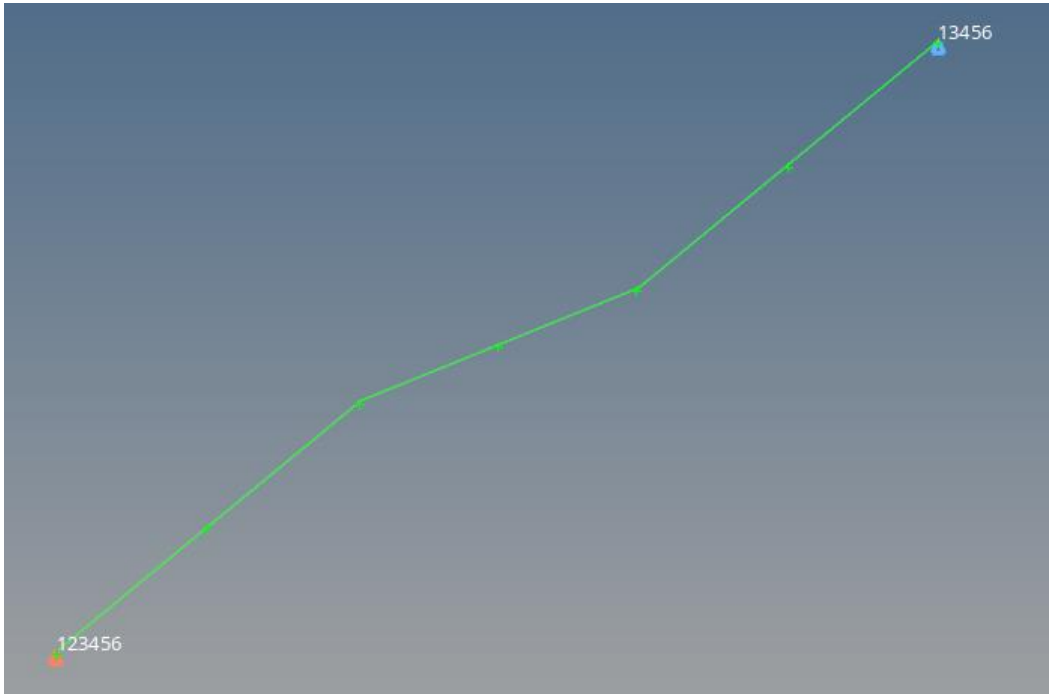
3 grnod_id 우클릭 하여 절점 set 생성
→ 고정 구속할 노드 추가





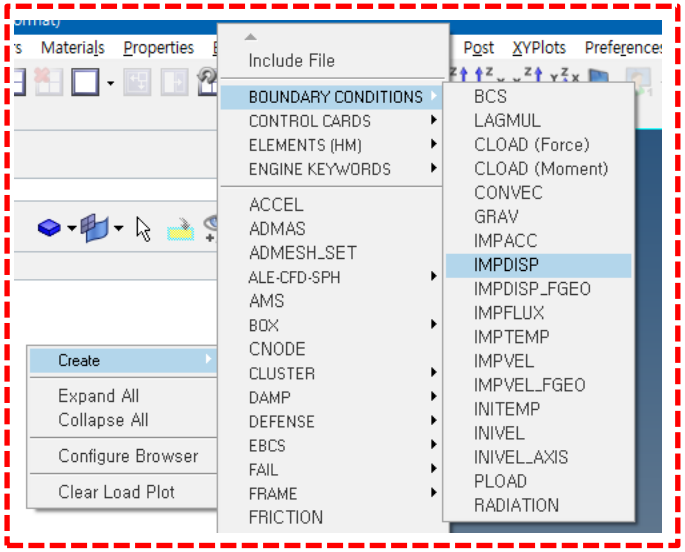
구속조건 설정 (2)

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	☐
Translation_Vx	☑
Translation_Vy	☐
Translation_Vz	☑
Rotation_Wx	☑
Rotation_Wy	☑
Rotation_Wz	☑



1 같은 방법으로 반대편 절점은 DOF2 제외하고 모두 구속

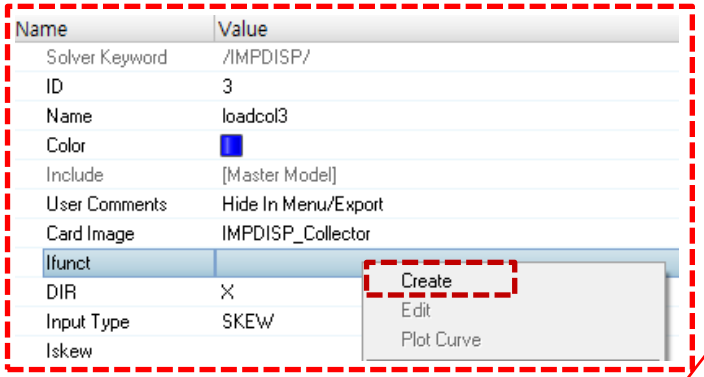
변위 조건 설정 (1)

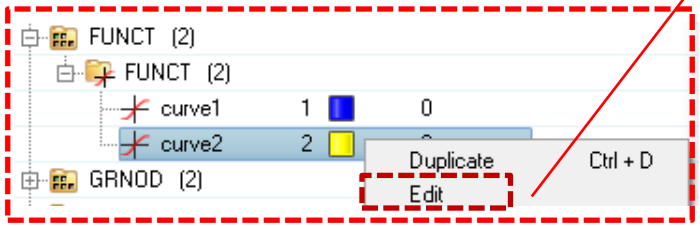


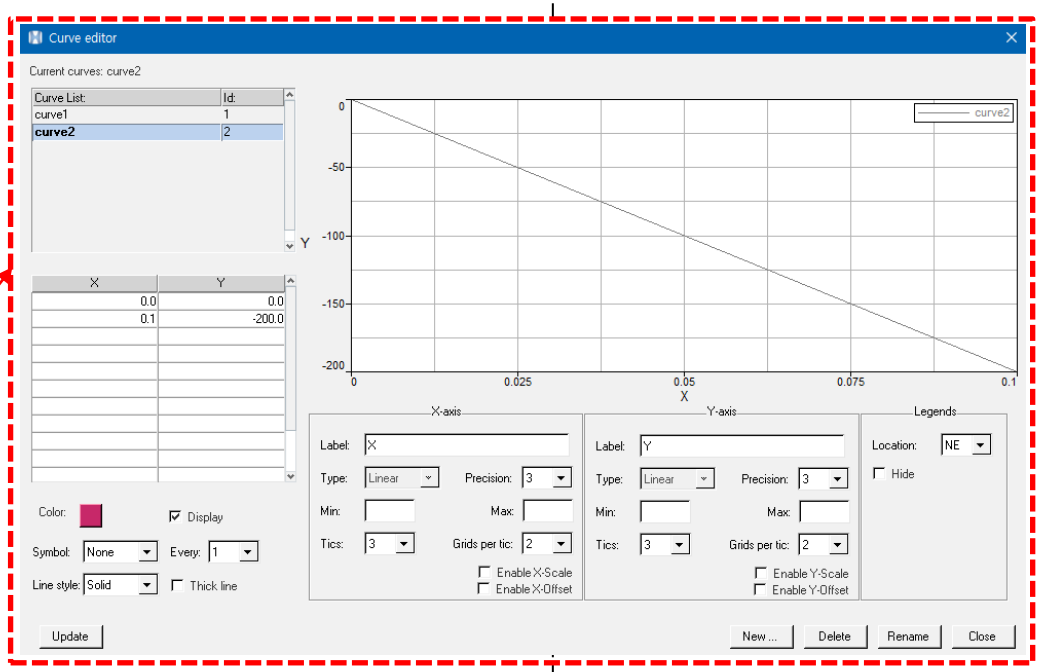
1 Solver탭 우클릭하여 IMPDISP 생성

2 lfunct 우클릭>Create
curve2 우클릭>Edit

시간-변위 함수 생성
(0,0) (0.1,-200)

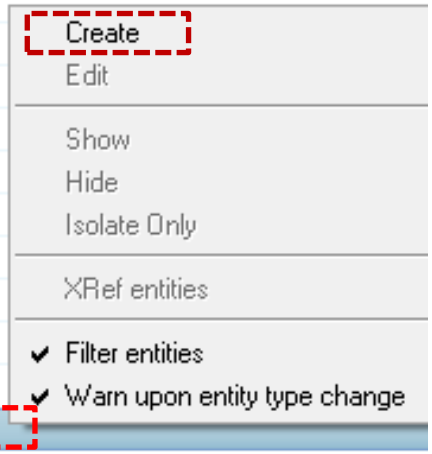


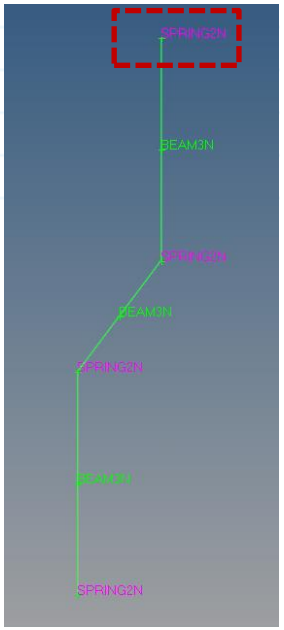




변위 조건 설정 (2)

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
Iskew	
Isensor	
Grnod_id	
I_coor	
scale_x	
scale_y	
Tstart	
Tstop	

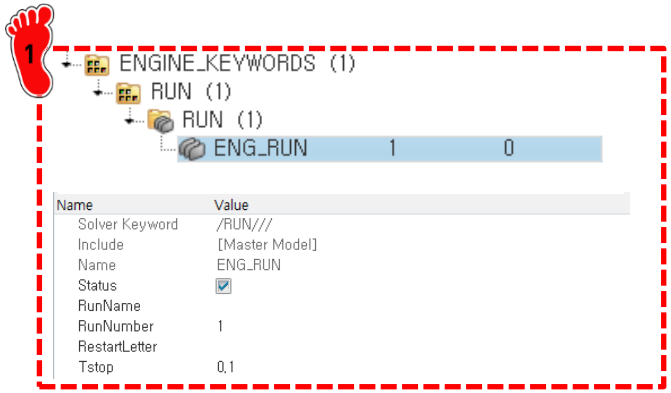


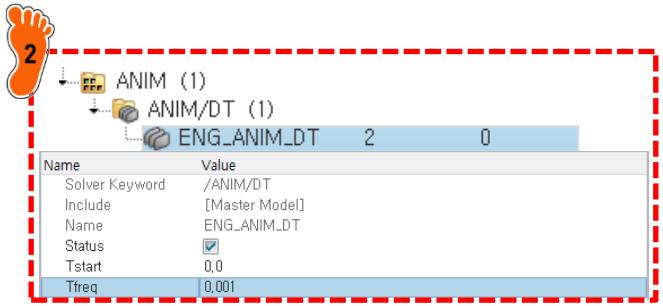


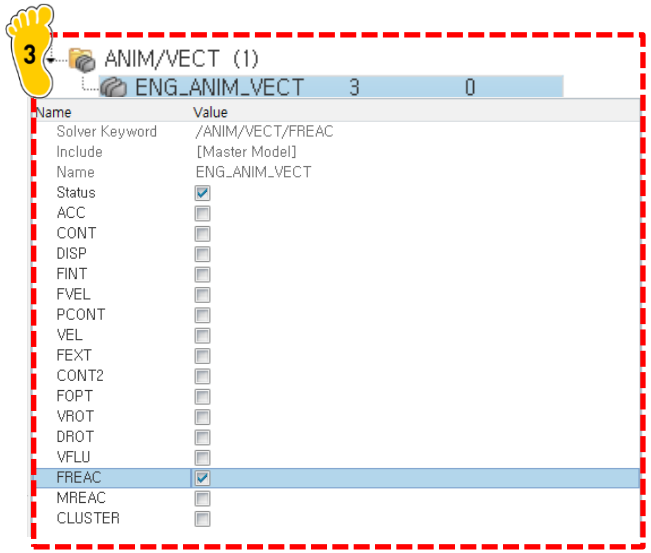
1 Grnod_id 클릭하여 set 생성 후 절점 추가 (상단 노드)

2 Dir은 Y로 설정

해석 케이스 설정







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

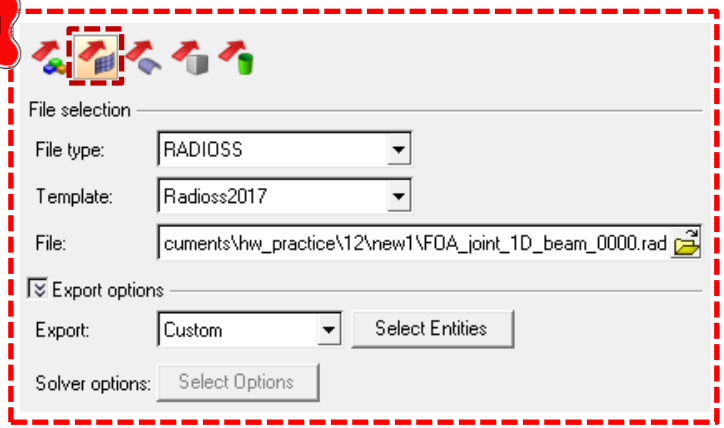
우클릭>Create>
ENGINE_KEYWORDS

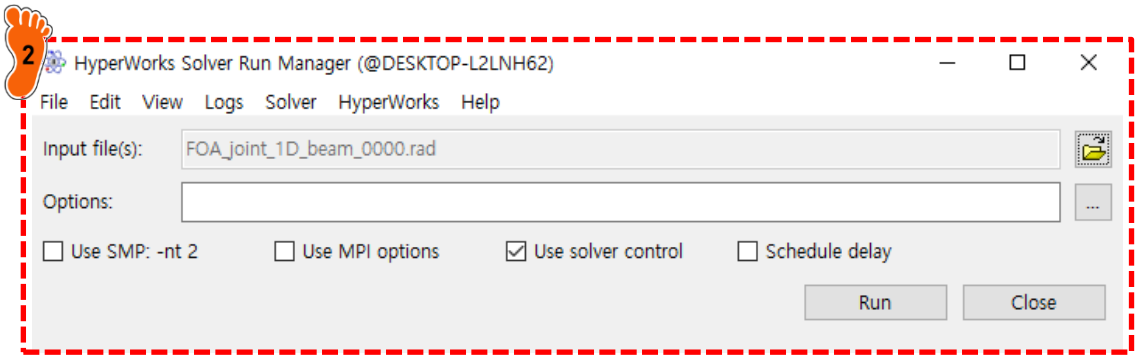
1 [ENG_RUN]
RunMumber: 1
Tstop: 0.1

2 [ENG_ANIM_DT]
Treq: 0.001

3 [ENG_ANIM_VECT]
FREAC

해석 실행





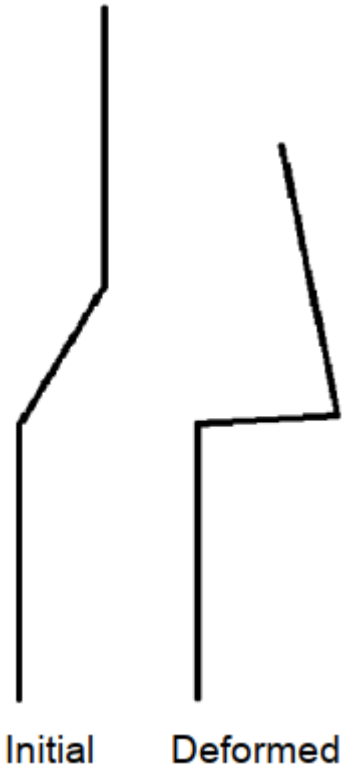
1 모델 export

2 Radioss에서 해석 실행

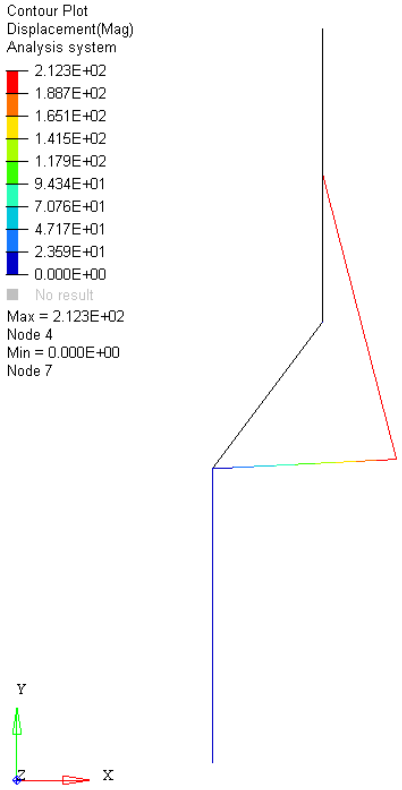
후처리

1 변형된 형상 확인

참고 논문 결과



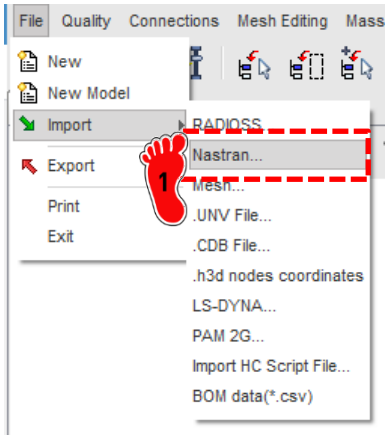
해석 결과

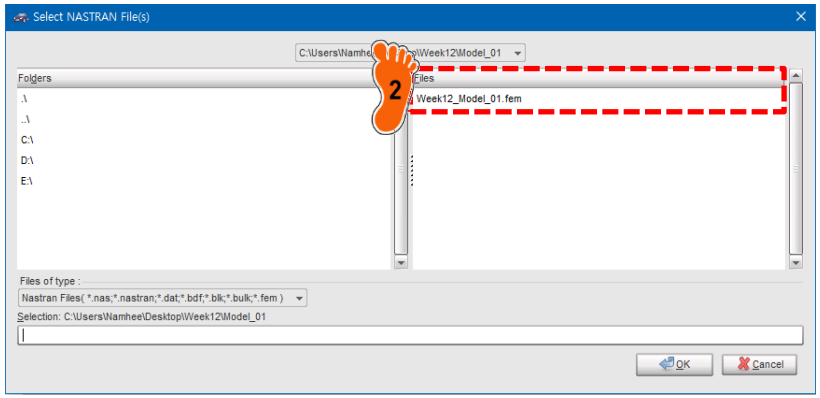


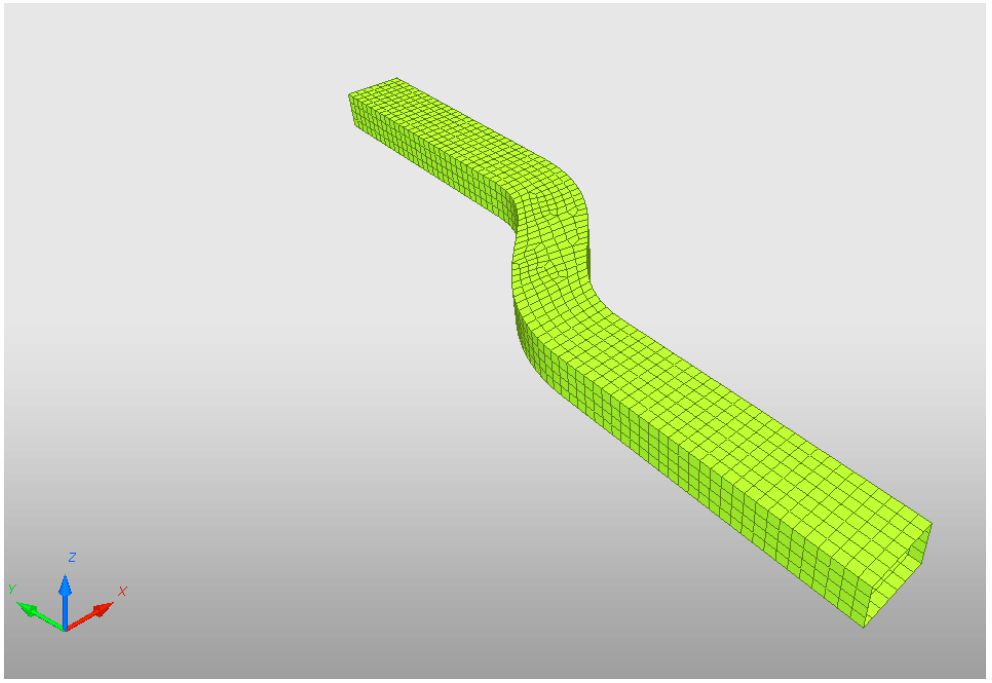
Computational time: 78.34 s

MID RAIL ANALYSIS 2 (SHELL ELEMENT)

모델 불러오기



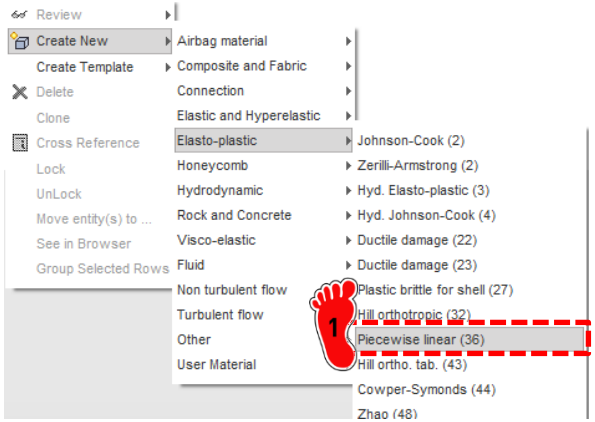


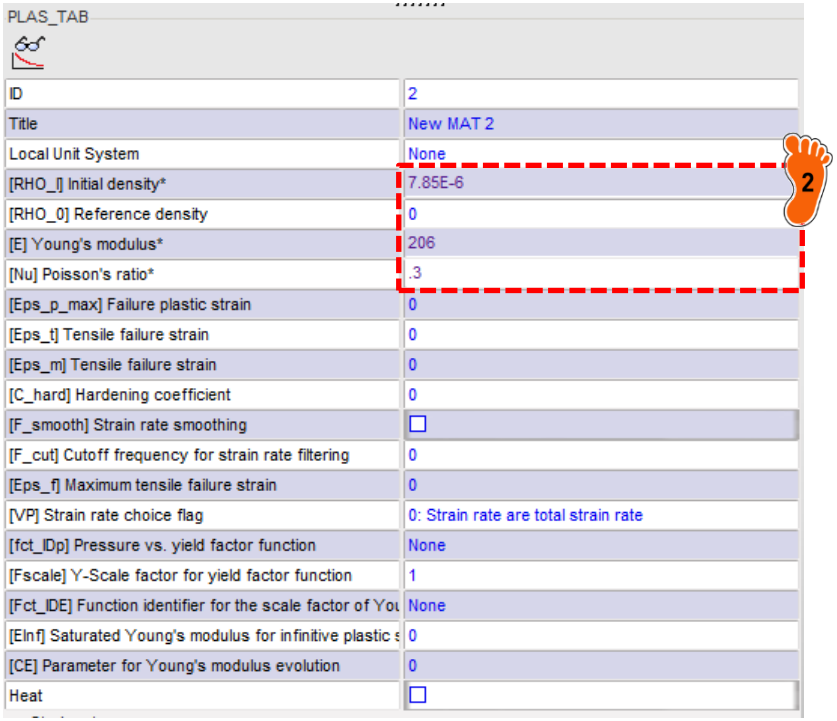


1 File > Import > Nastran 클릭

2 모델 파일 클릭 후 불러오기 (Week12_Model_02.fem)

재료 물성 입력 (1)

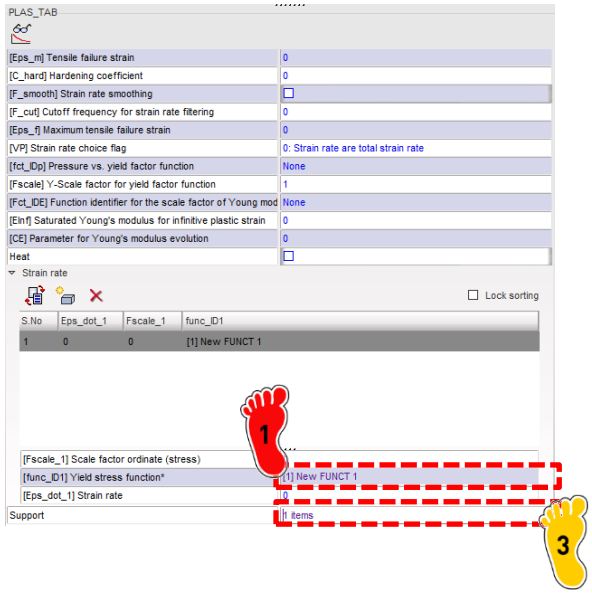


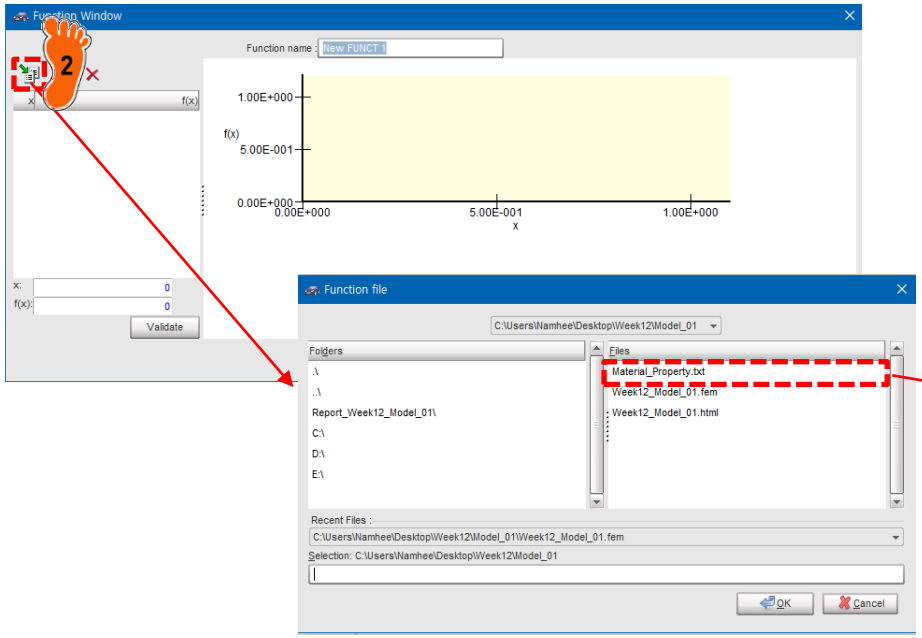


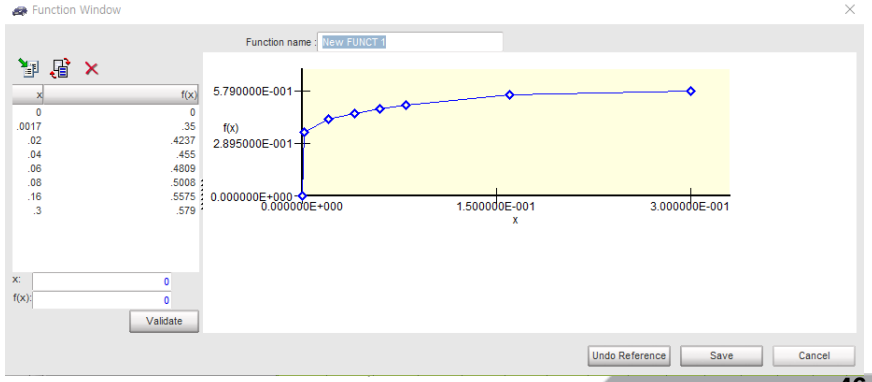
1 Model > Material 클릭, 마우스 우클릭 후 Create New > Piecewise linear(36) 재료 생성

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

재료 물성 입력 (2)

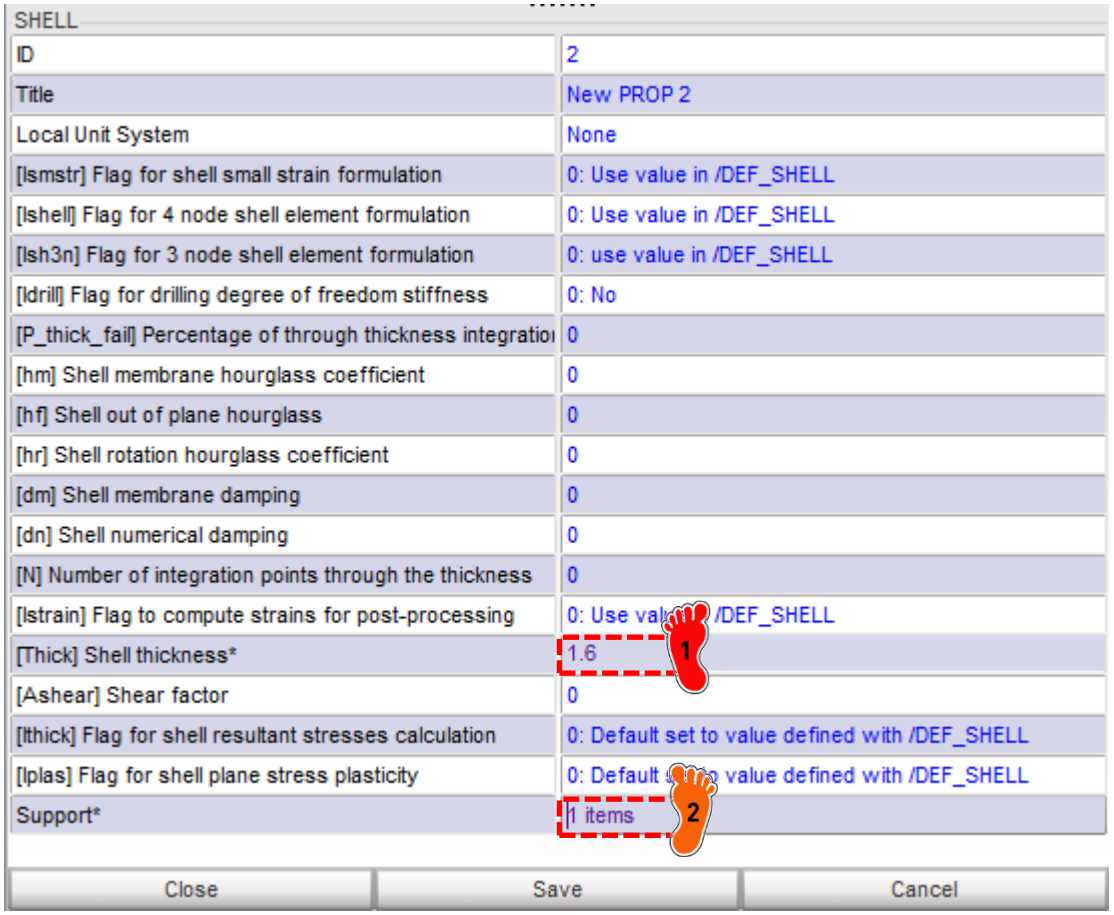






1. Stress-Strain curve 입력 (마우스 우클릭 후 "Define function" 클릭)
2. "Import curve" 클릭 후 "Material Property.txt" 파일 불러오기
3. 모델에 재료 적용

특성 입력

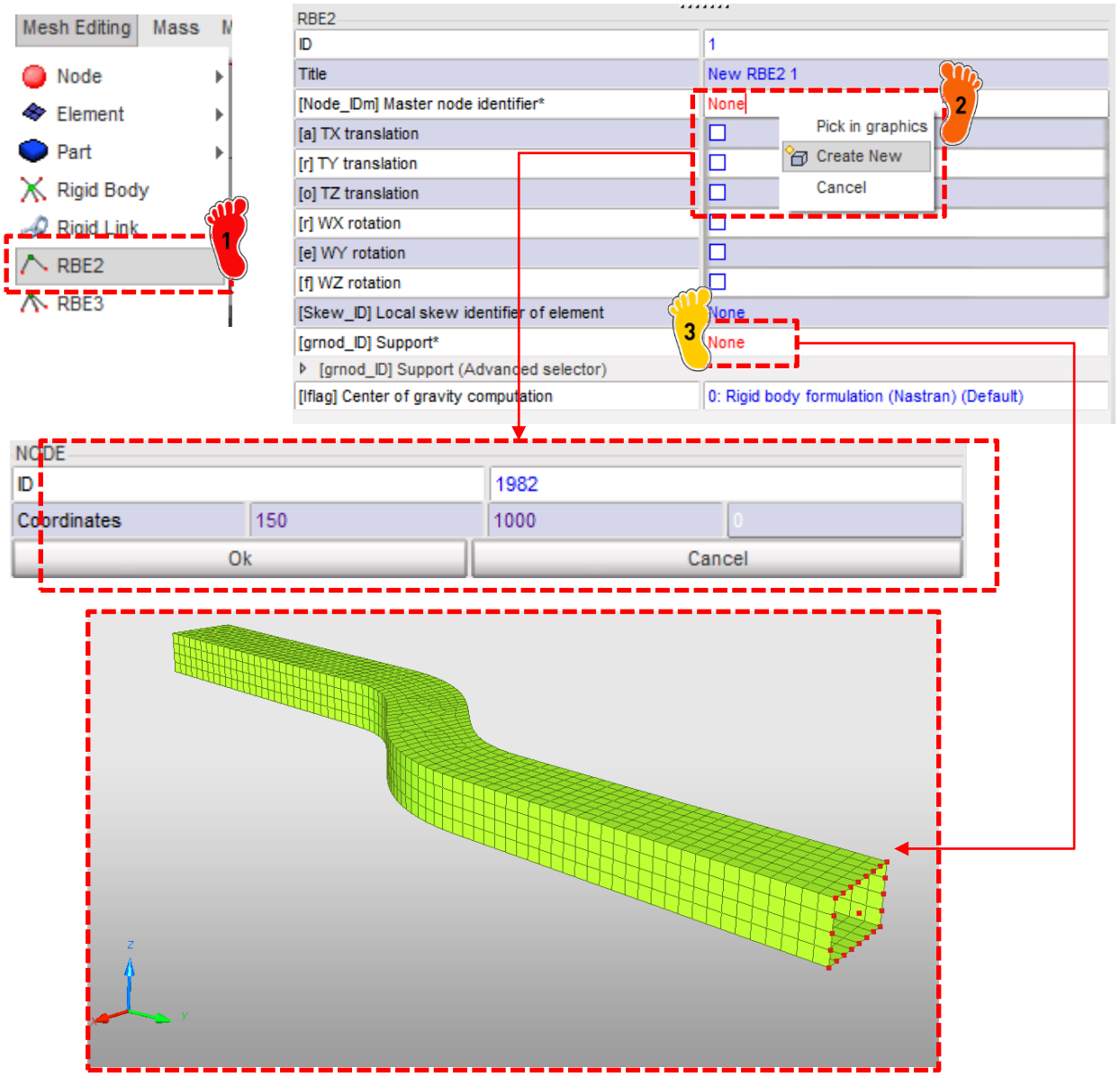


1. Model > Property 클릭, 마우스 우클릭 후 Create New > Surface > Shell (1) 클릭

   두께 1.6mm 입력

2. 모델에 특성 적용

RBE2 생성

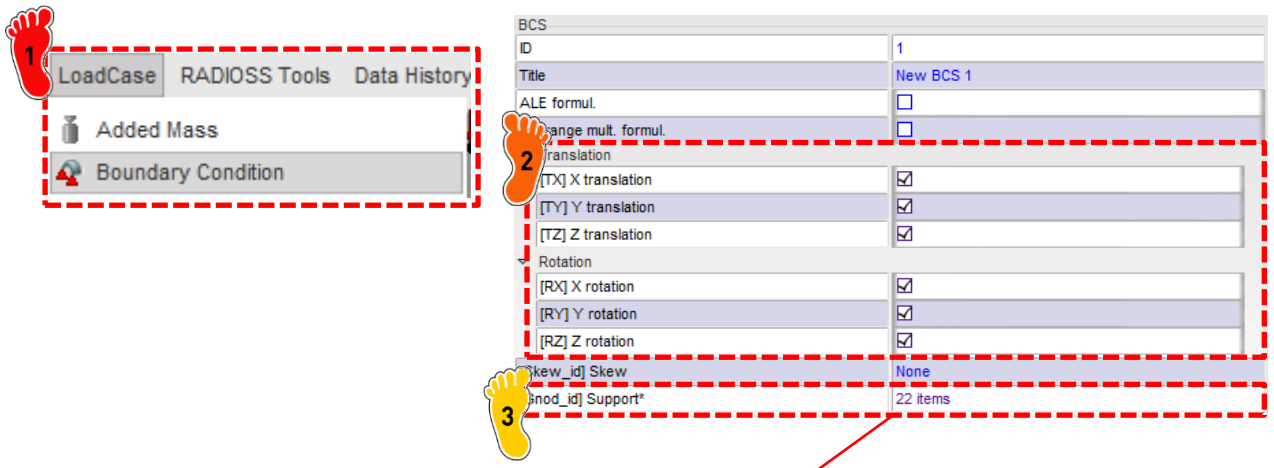


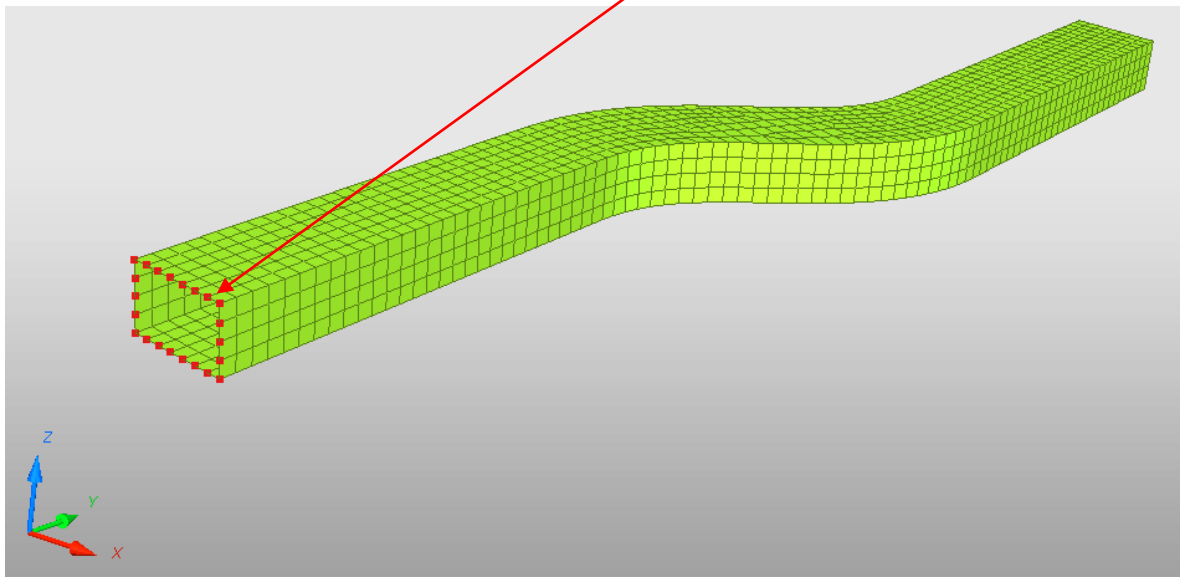
1. Mesh Editing > RBE2 생성 마우스 우클릭 후 Create New 클릭

2. Master node 좌표: (150, 1000, 0)

3. Support: Y방향 끝단 노드 선택

구속조건 설정 (1)



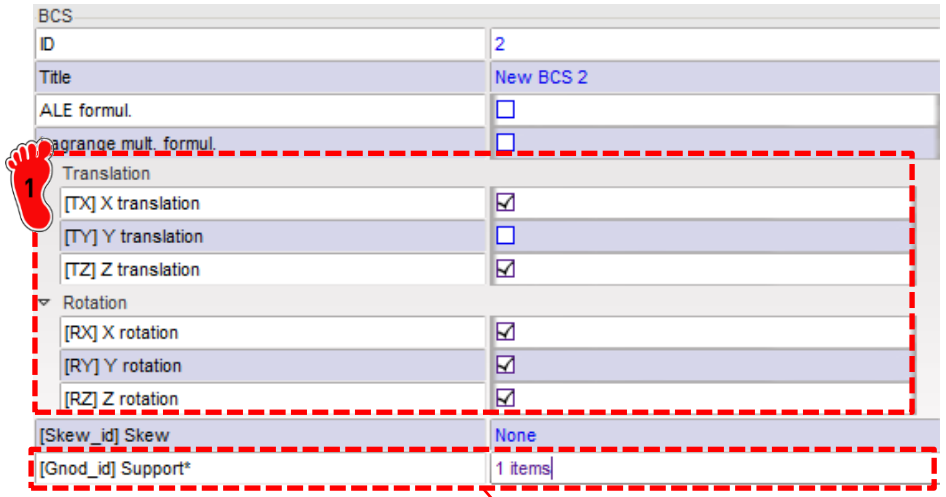


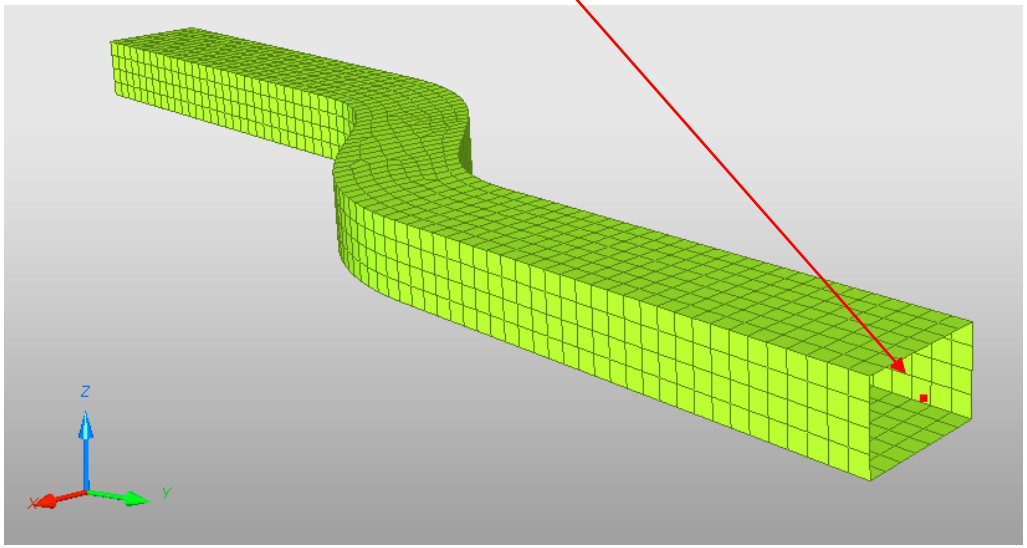
1 LoadCase > Boundary Condition 클릭, 마우스 우클릭 후 Create new 클릭

2 완전 구속 설정

3 구속할 절점 선택

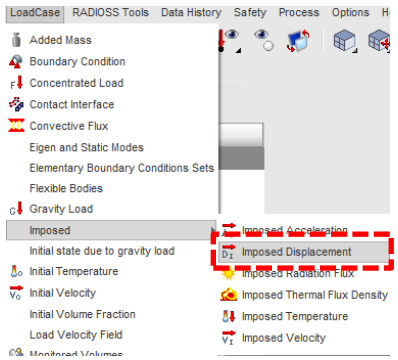
구속조건 설정 (2)

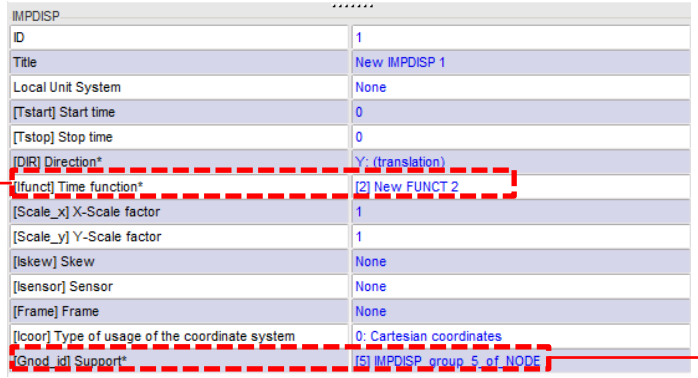


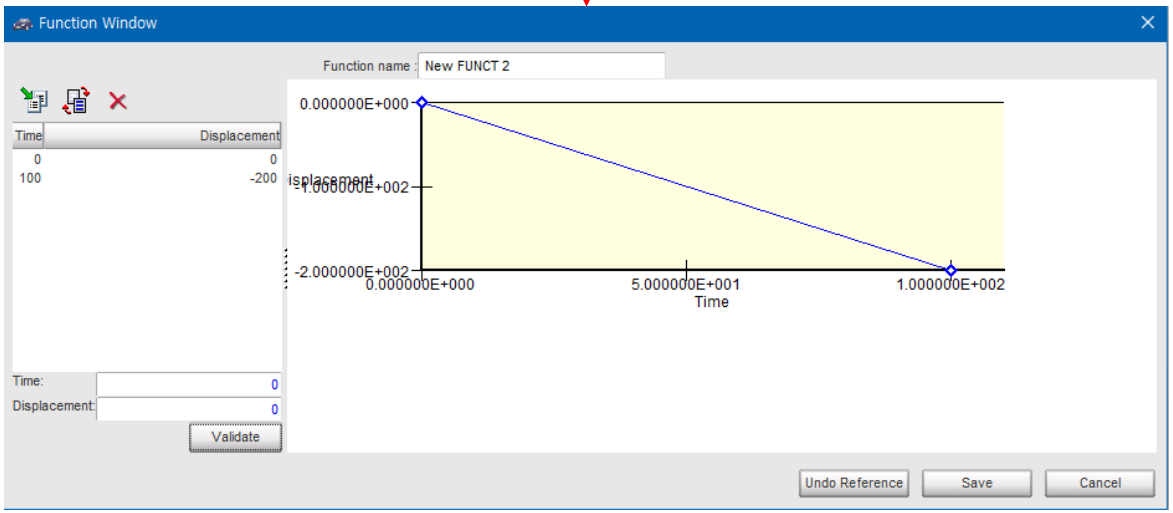


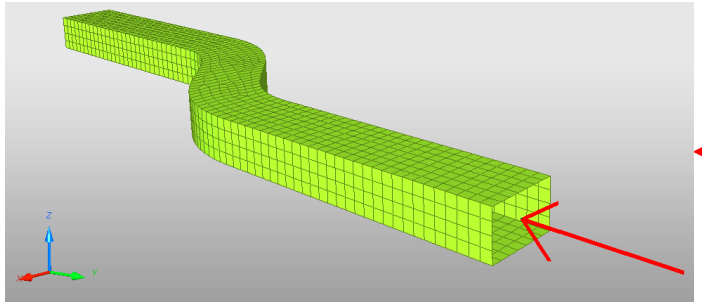
강체 중심 절점은 Y 방향 구속조건을 제외한 모든 자유도 구속
(Tx,Tz,Rx,Ry,Rz 구속)

변위조건 설정









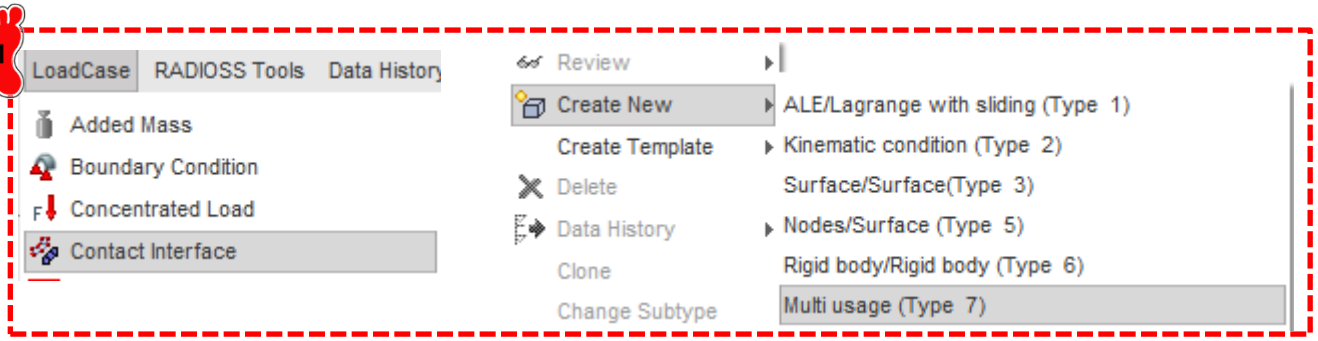
1. LoadCase > Imposed > Imposed Displacement 클릭,
마우스 우클릭 후
Imposed displacements 생성

2. 시간-변위 그래프 생성
(0,0), (100,-200)

DIR: Y

3. Support: 앞서 만든 RBE2 요소의 master node 선택

접촉조건 설정



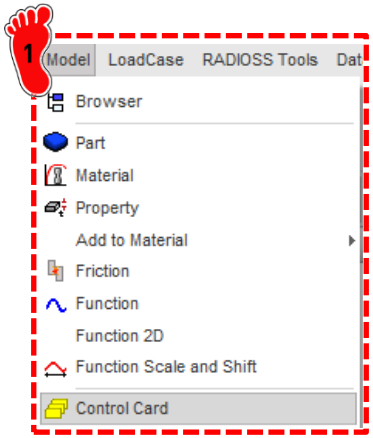
1 LoadCase > Contact Interface클릭, 마우스 우클릭 후 "Multi usage (Type 7) 클릭

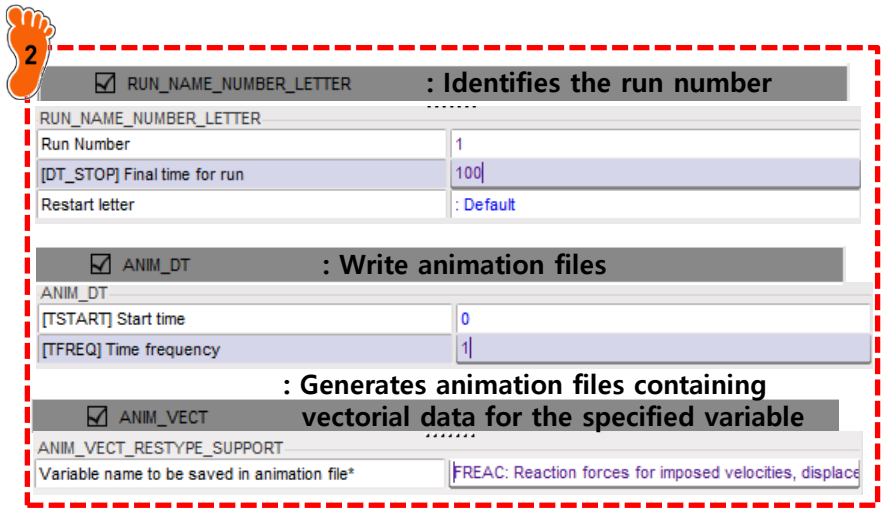
Interface stiffness scale factor
(1.0 real: contact stiffness is taken from the master segment)

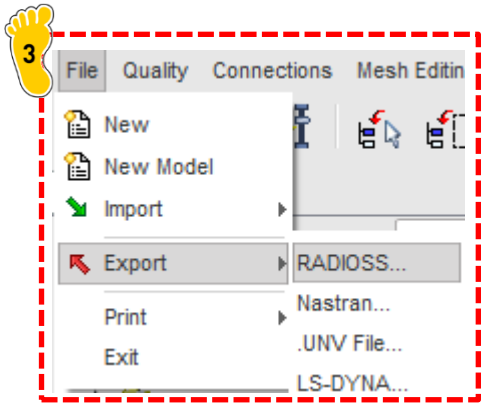
Minimum gap for activation of interface

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

해석 케이스 설정







1 Model > Control Card클릭

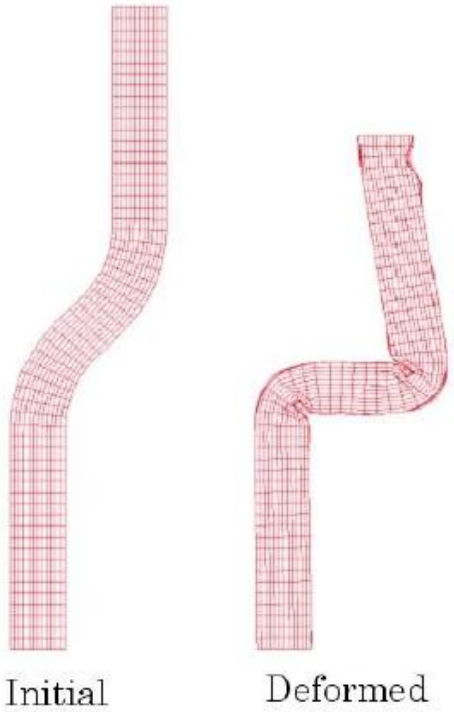
2 파라미터 설정

3 모델 export (RADIOSS 선택)

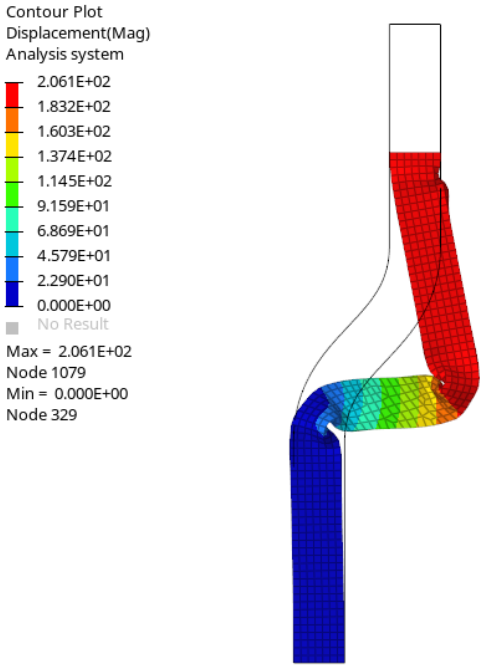
후처리 (1)

1 변형된 형상 확인

참고 논문 결과

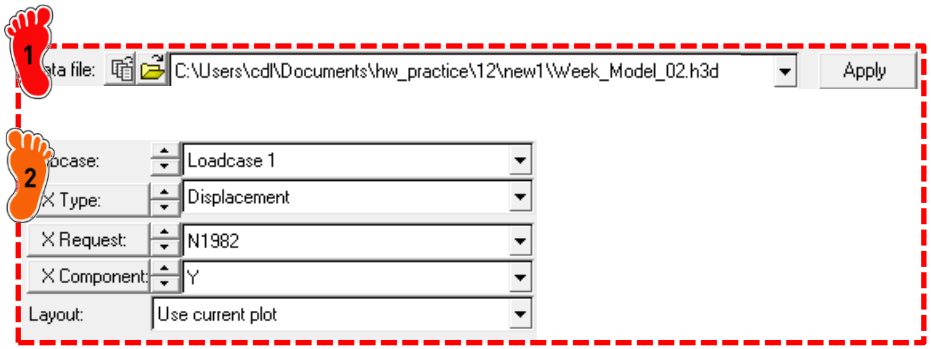


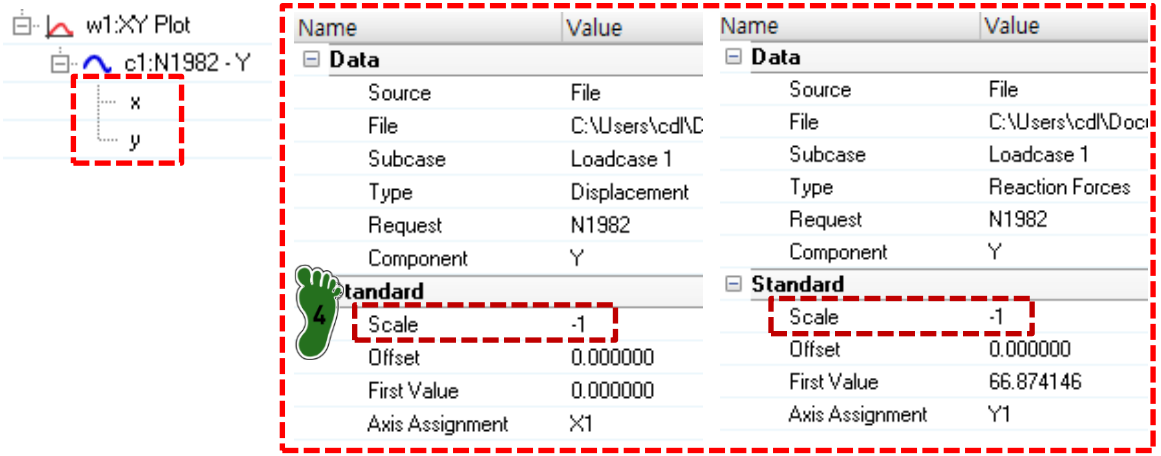
해석 결과



Computational time: 87.70 s

후처리 (2)





1 HyperGraph 2D > open H3D 파일

2 X축 > Displacement, N1982, Y 설정

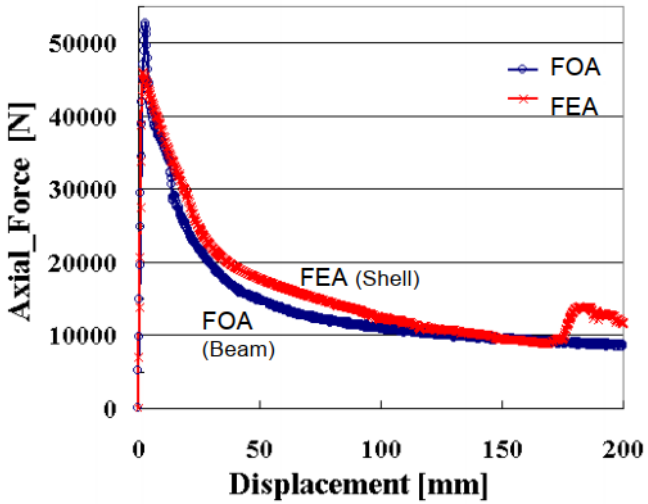
3 Y축 > Reaction Forces, N1982, Y 설정

4 X, Y축 Scale > -1 설정 (양수로 전환)

후처리 (2)

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

참고 논문 결과



해석 결과