Crashworthiness

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

한양대학교 HANYANG UNIVERSITY

CDL Computational Design Lab

목차

- 예제 문제
  - Non-linear joint moment analysis
  - Mid rail analysis
- 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

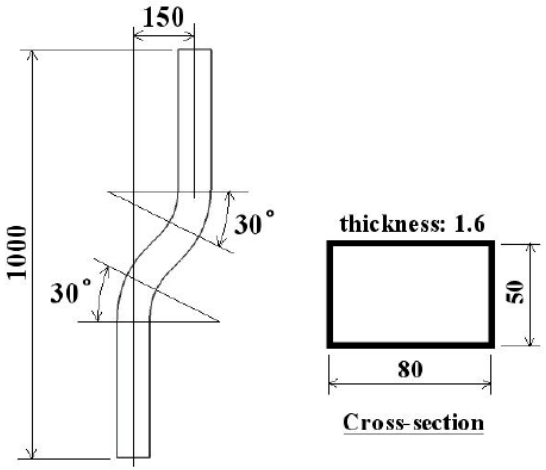
2004-01-1660

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

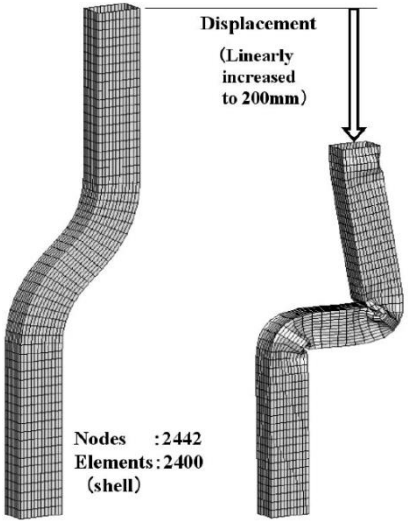
Hidekazu Nishigaki
Toyota Central R&D Labs., Inc.

Noboru Kikuchi
The University of Michigan

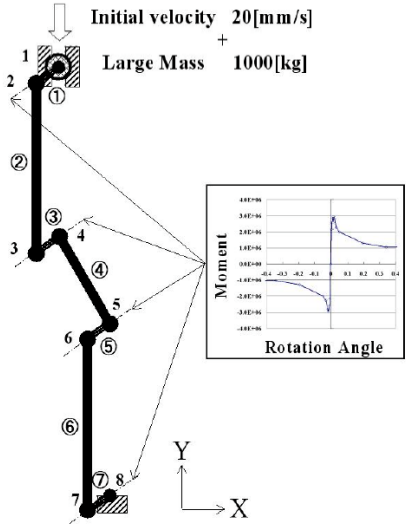
기하 형상 정보



쉘 모델 해석



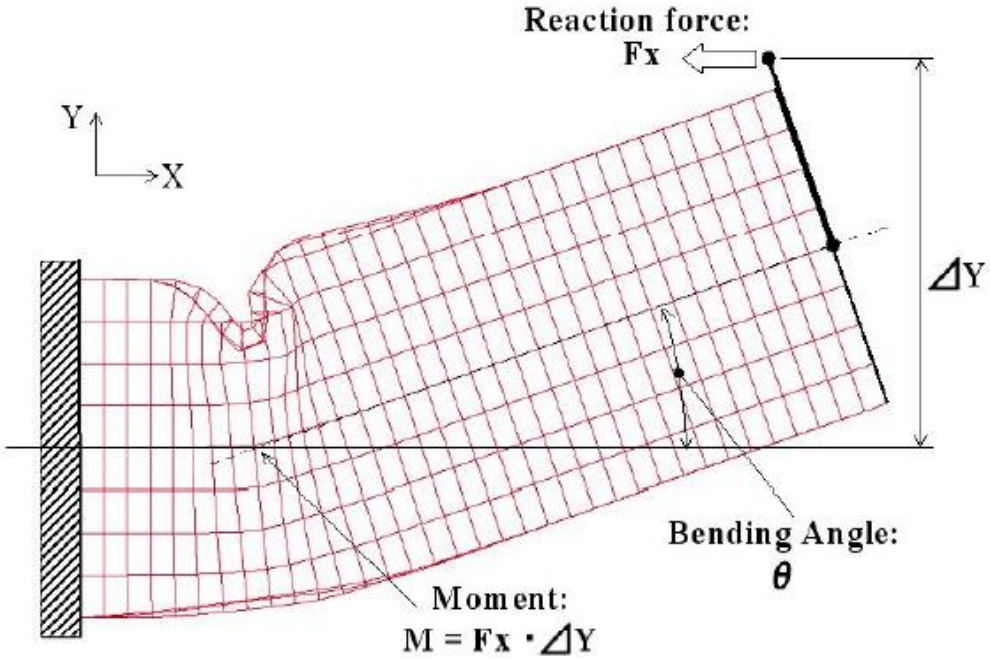
빔 모델 해석

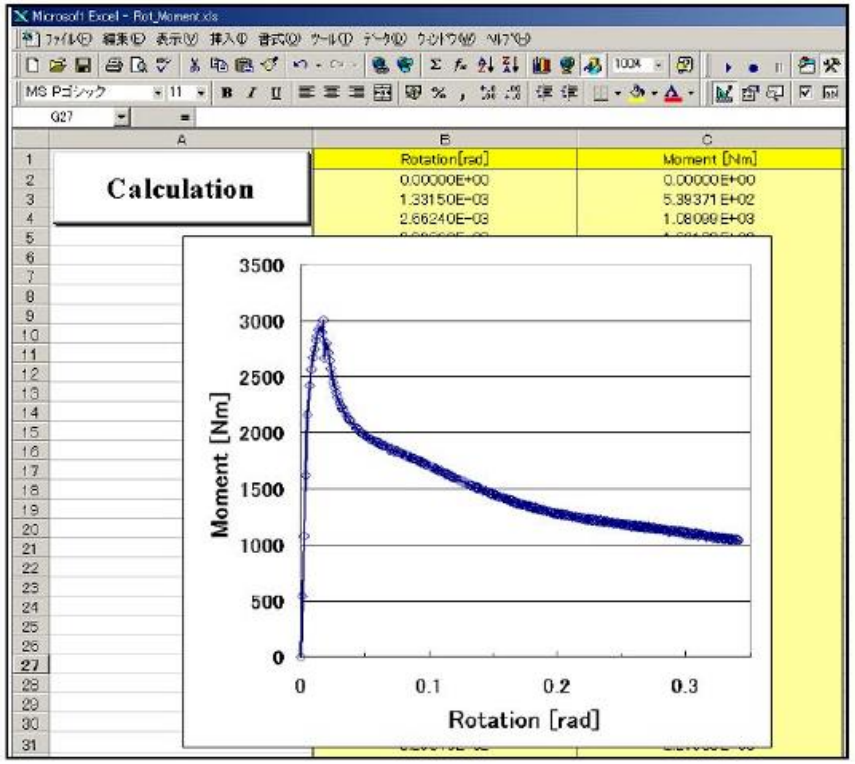


JOINT MOMENT ANALYSIS

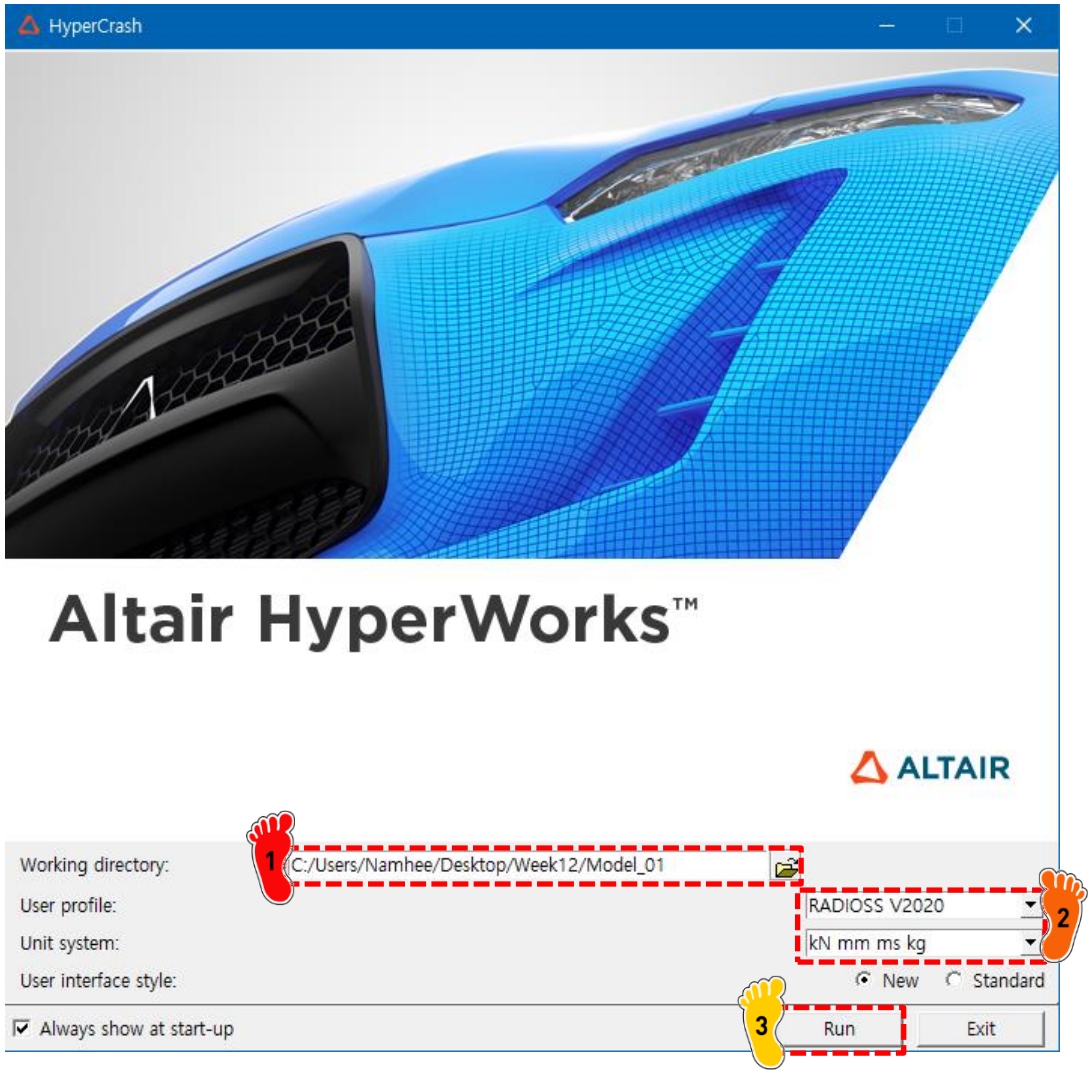
예제: JOINT MOMENT ANALYSIS

- Shell 모델의 등가 강성 계산



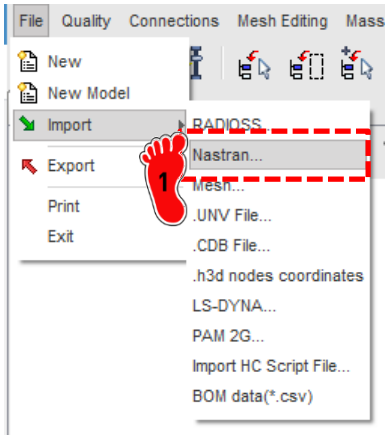


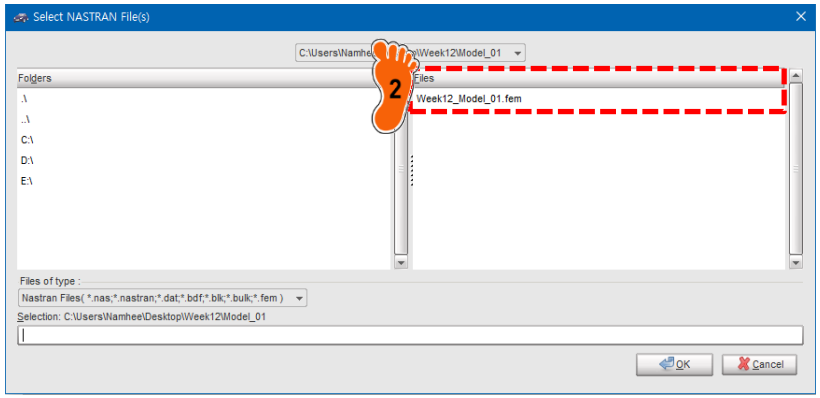
HYPERCRASH 실행

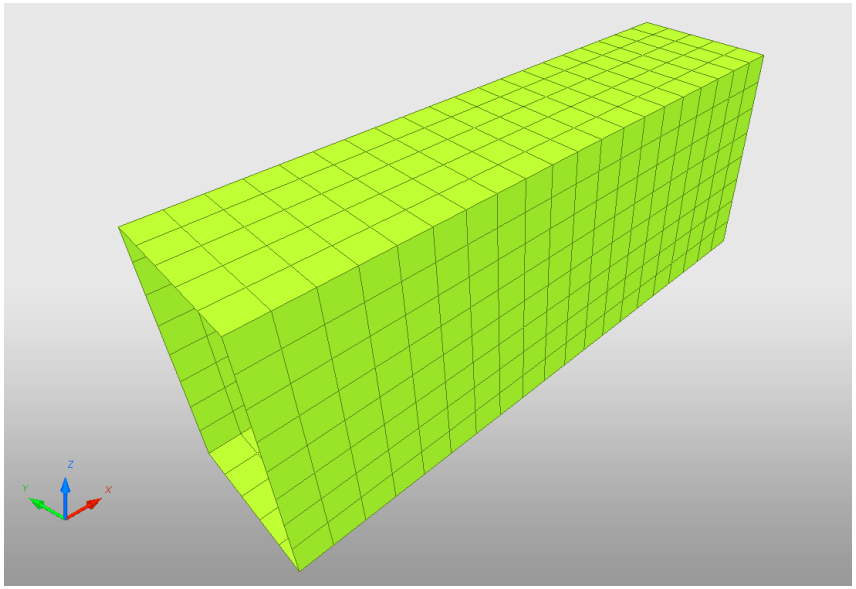


1. HyperCrash2020 실행 후 Working directory 설정 (모델 파일이 있는 폴더로 지정)
2. User profile, Unit system 설정
3. Run 클릭

유한요소해석 모델 불러오기



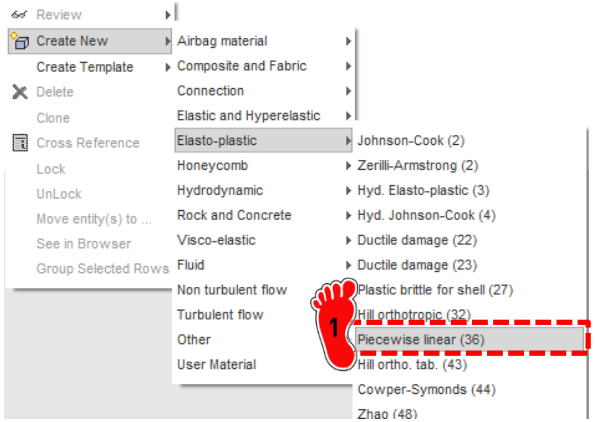


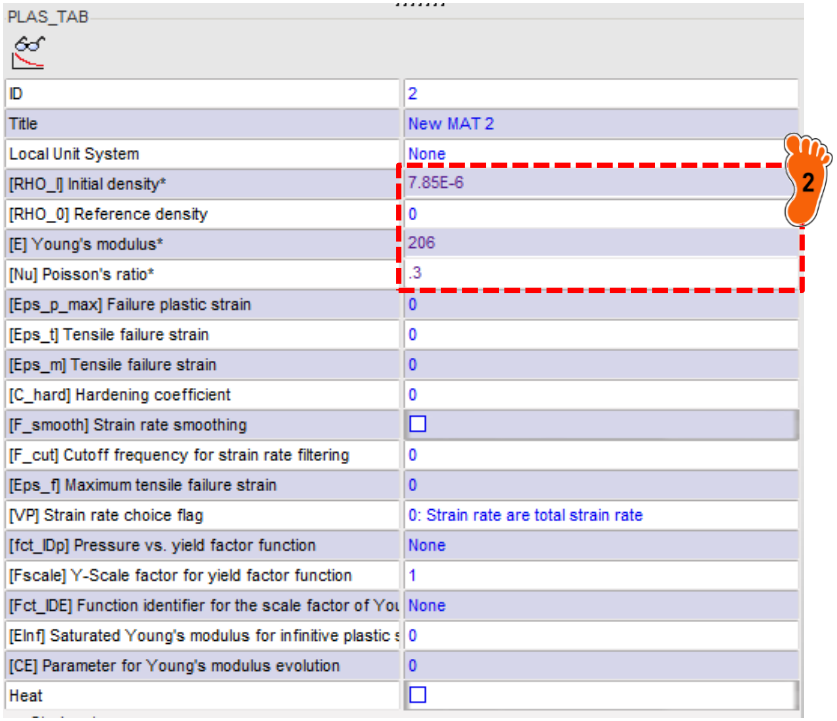


1 File > Import > Nastran 클릭

2 모델 파일 클릭 후 불러오기 (Week12_Model_01.fem)

재료 물성 입력 (1)

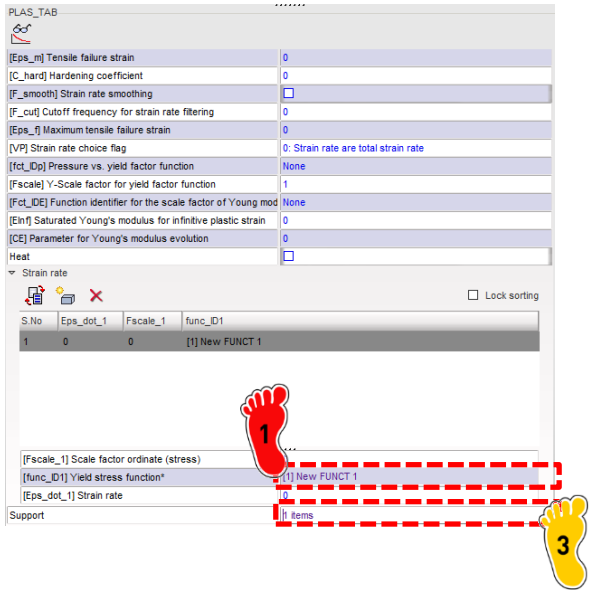


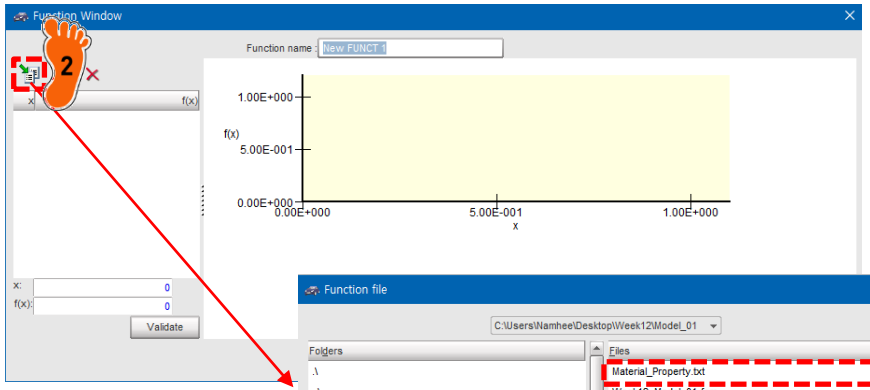


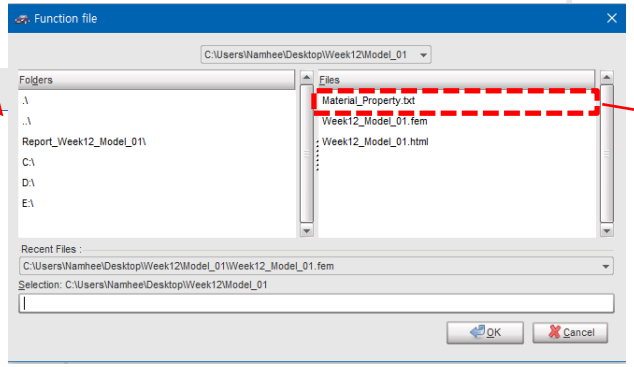
1 Piecewise linear(36) 재료 생성

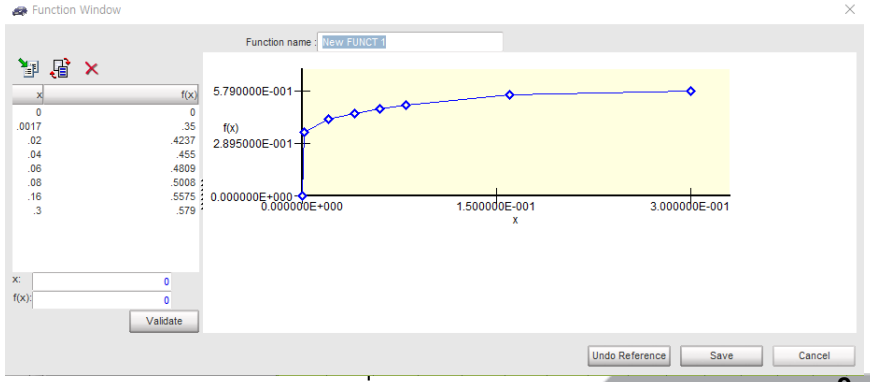
2 재료 물성치 입력
[RHO_I]: 7.85e-6
[E]: 240
[Nu]: 0.3

재료 물성 입력 (2)



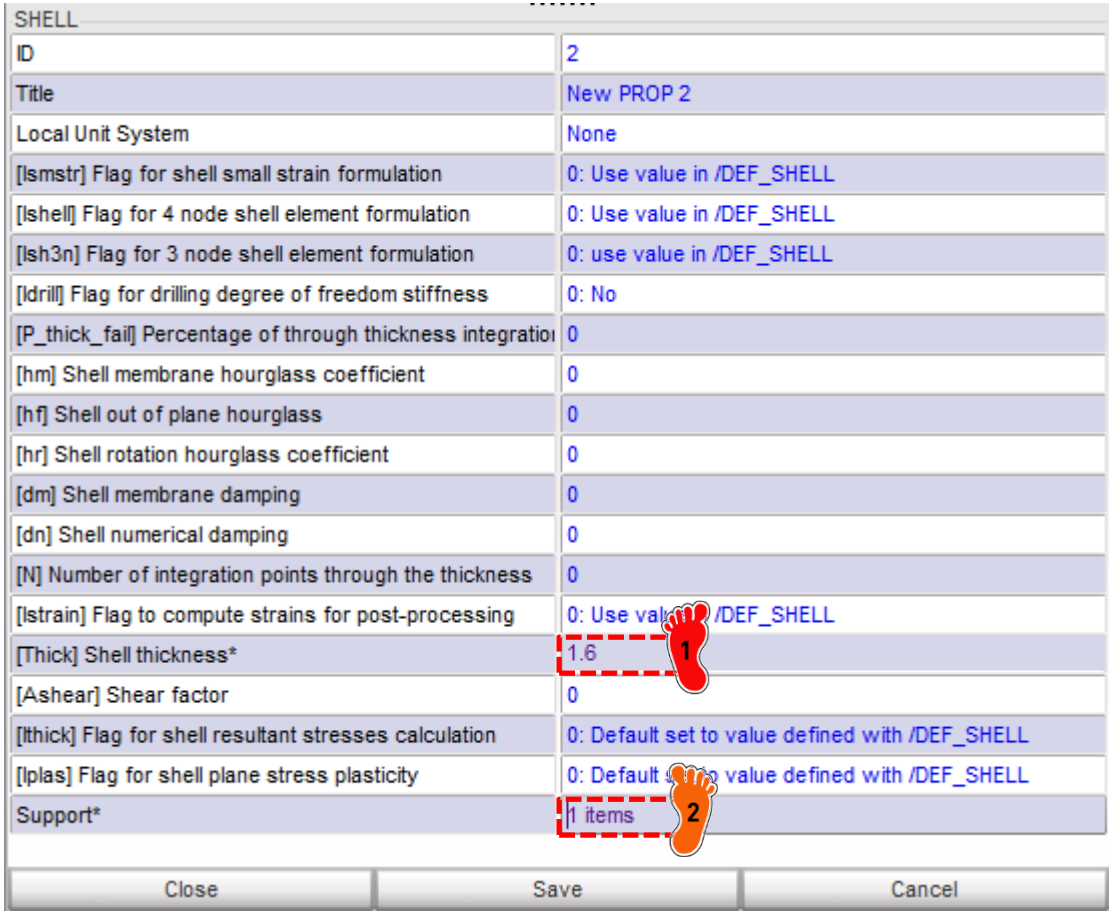






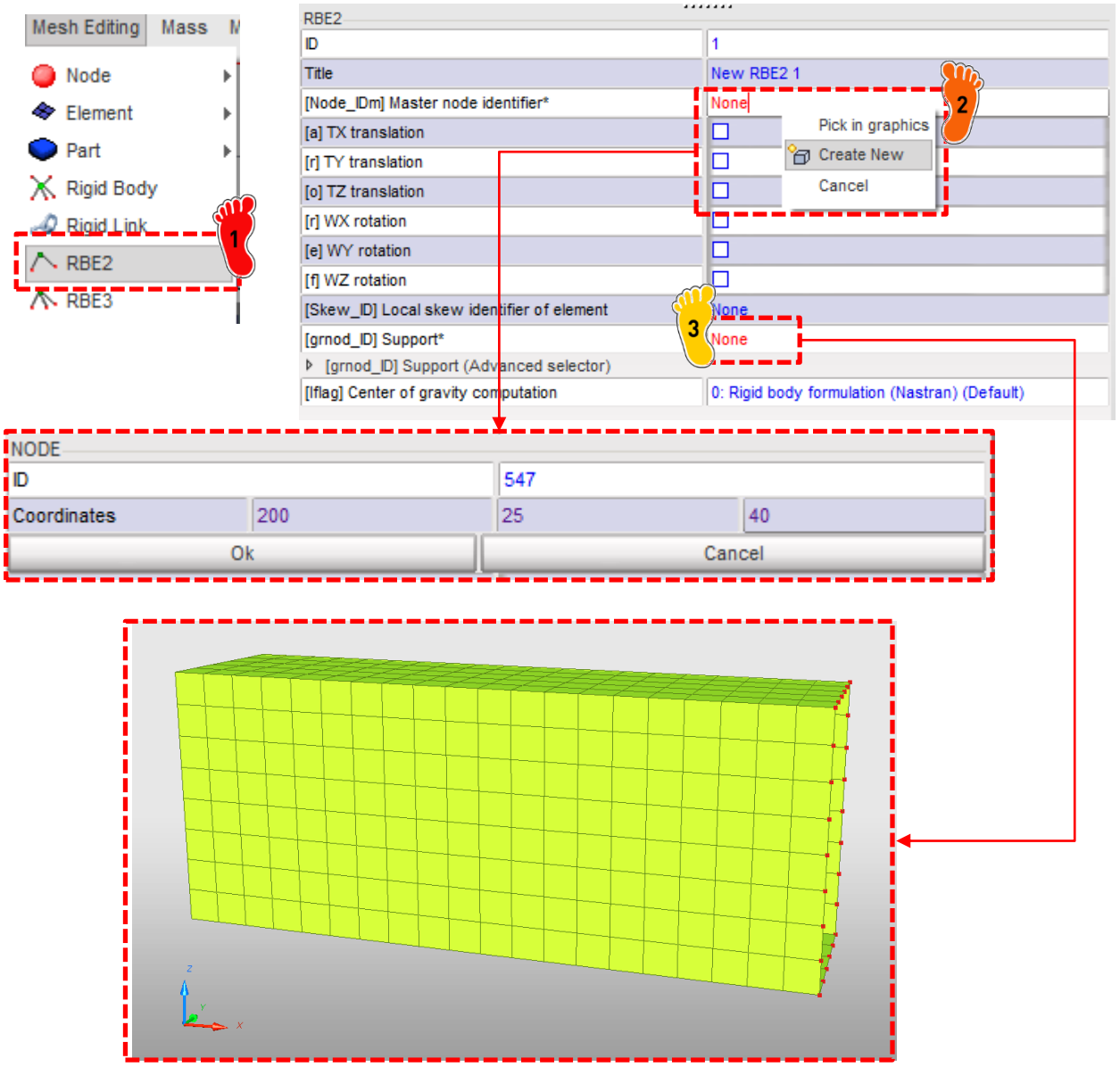
1. Stress-Strain curve 입력 (마우스 우클릭 후 "Define function" 클릭)
2. "Import curve" 클릭 후 "Material Property.txt" 파일 불러오기
3. 모델에 재료 적용

특성 입력



1. 쉘 property 생성 후 두께 1.6mm 입력
2. 모델에 특성 적용

RBE2 생성 (1)

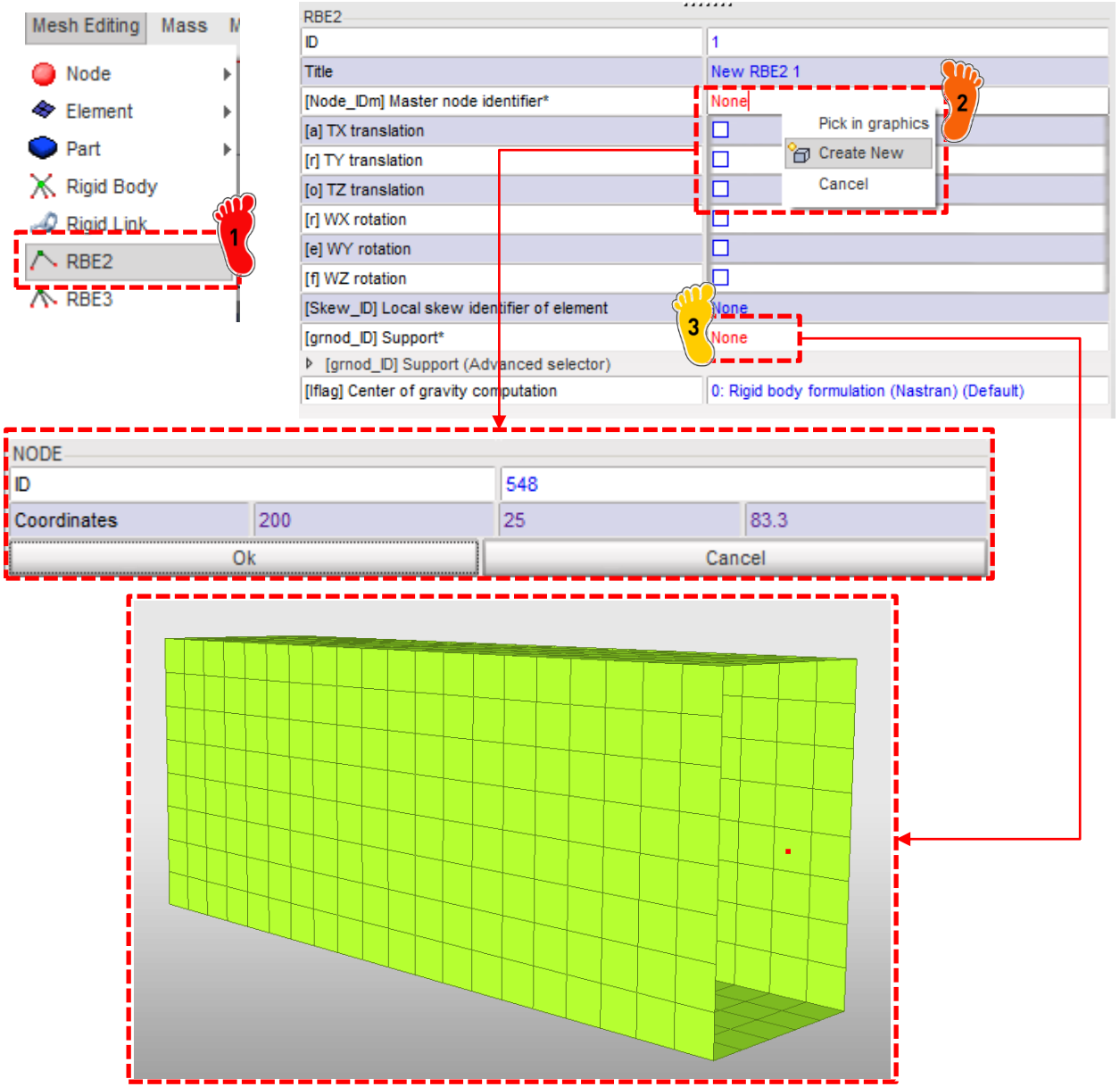


1 RBE2 생성

2 Master node 좌표: (200, 25, 40)

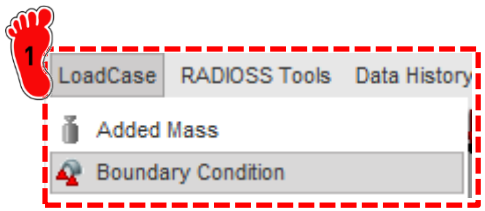
3 Support: X방향 끝단 노드 선택

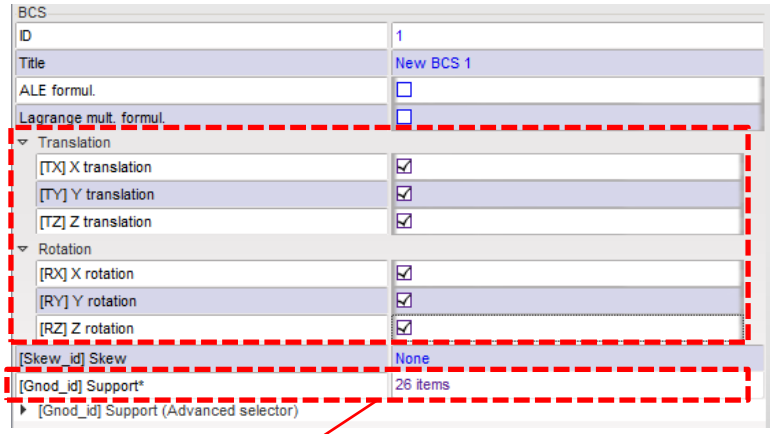
RBE2 생성 (2)

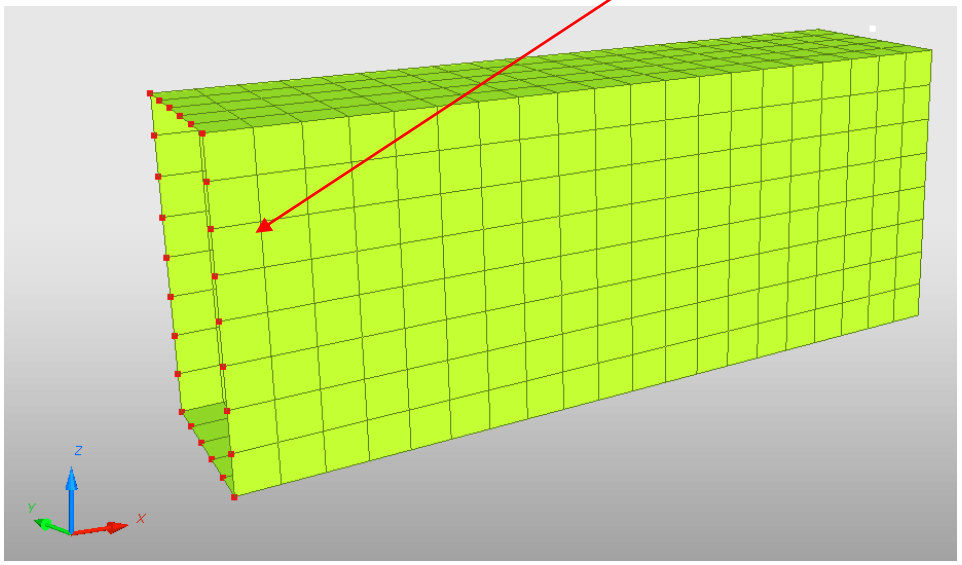


1. 새로운 RBE2 생성
2. Master node 좌표: (200, 25, 83.3)
3. Support: 이전에 만든 RBE2 mater node

구속조건 설정





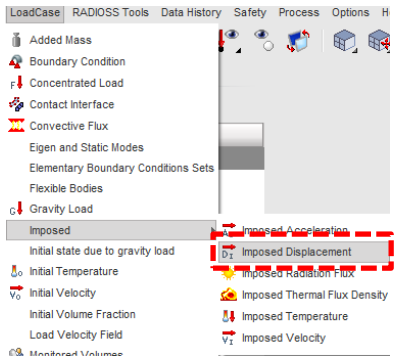


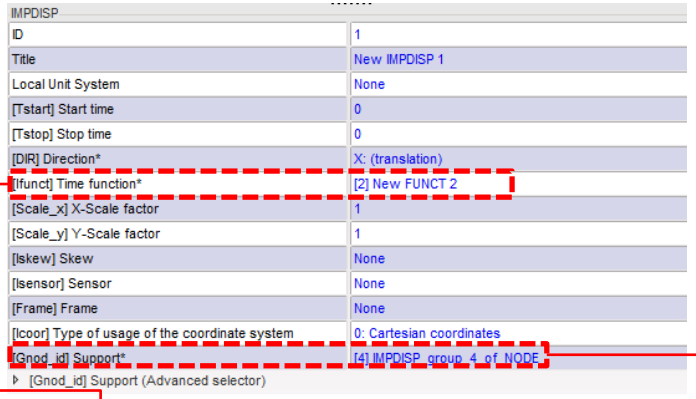
1 Boundary condition 생성

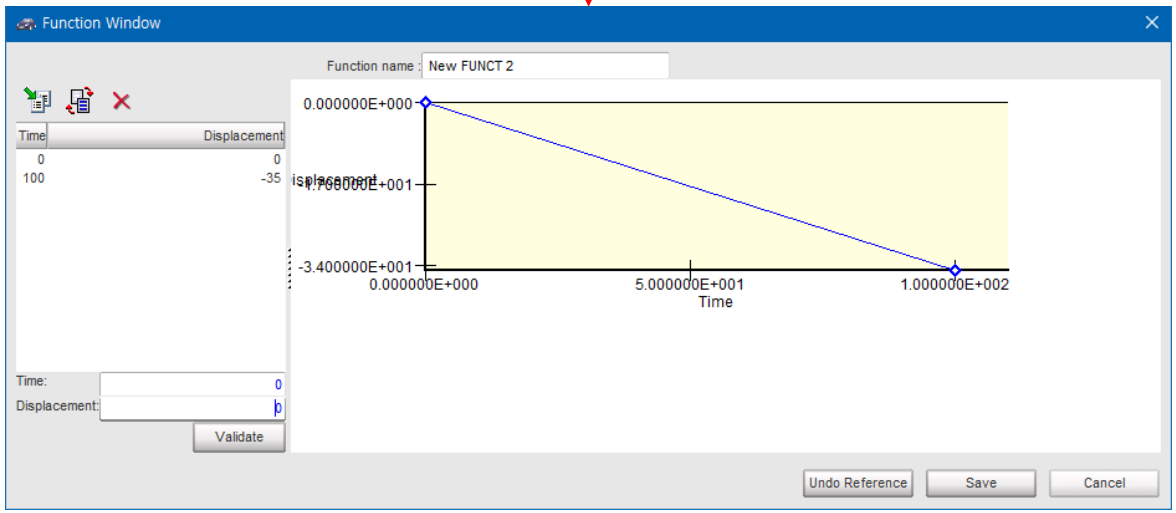
2 완전 구속 설정

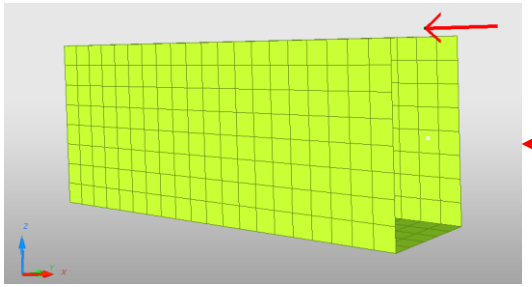
3 구속할 절점 선택

변위조건 설정



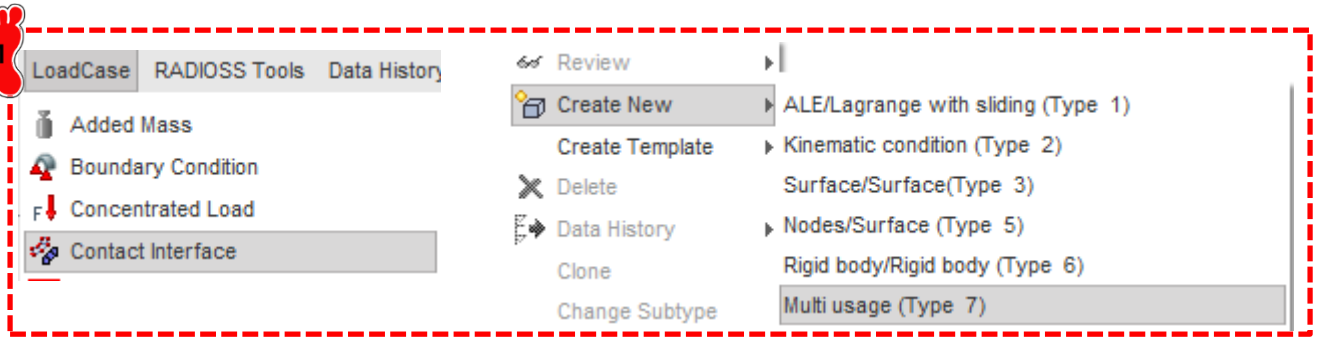






1. Imposed displacement 생성
2. 시간-변위 그래프 생성 (0,0), (100,-35)
3. Support: 앞서 만든 RBE2 요소의 master node 선택

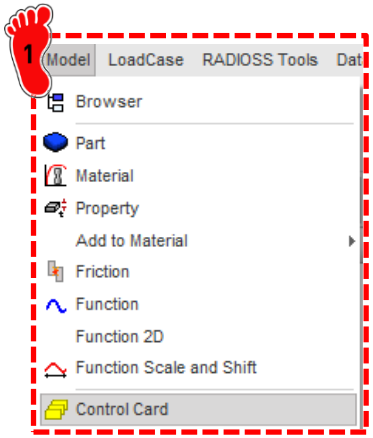
접촉조건 설정

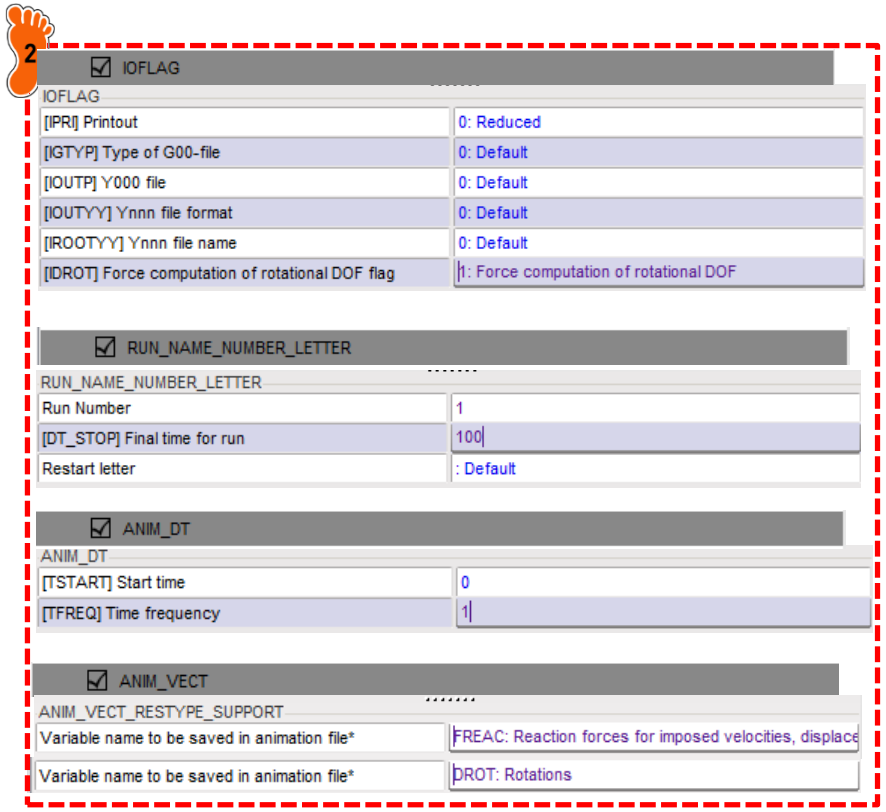


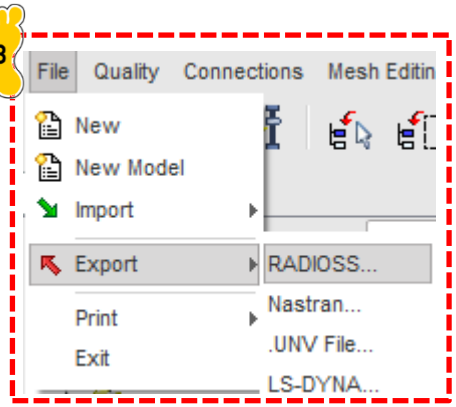
1 LoadCase > Contact Interface클릭,
마우스 우클릭 후 "Multi usage (Type 7) 클릭

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

해석 케이스 설정







1 Model > Control Card클릭

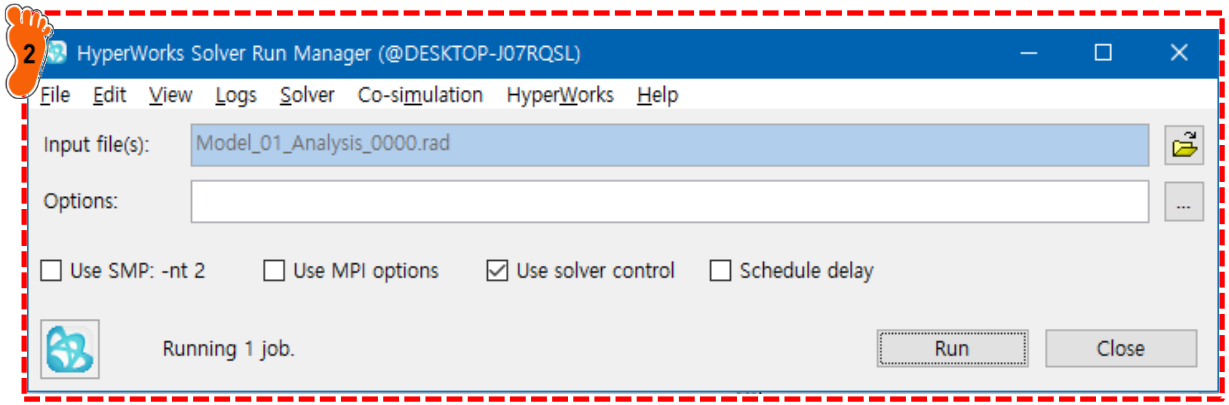
2 파라미터 설정

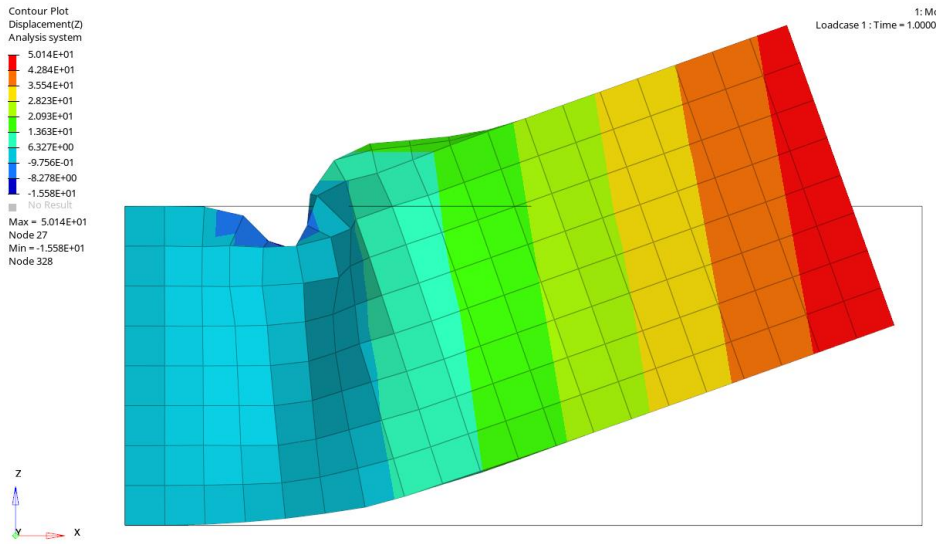
3 모델 export (RADIOSS 선택)

해석 실행 (RADIOSS)

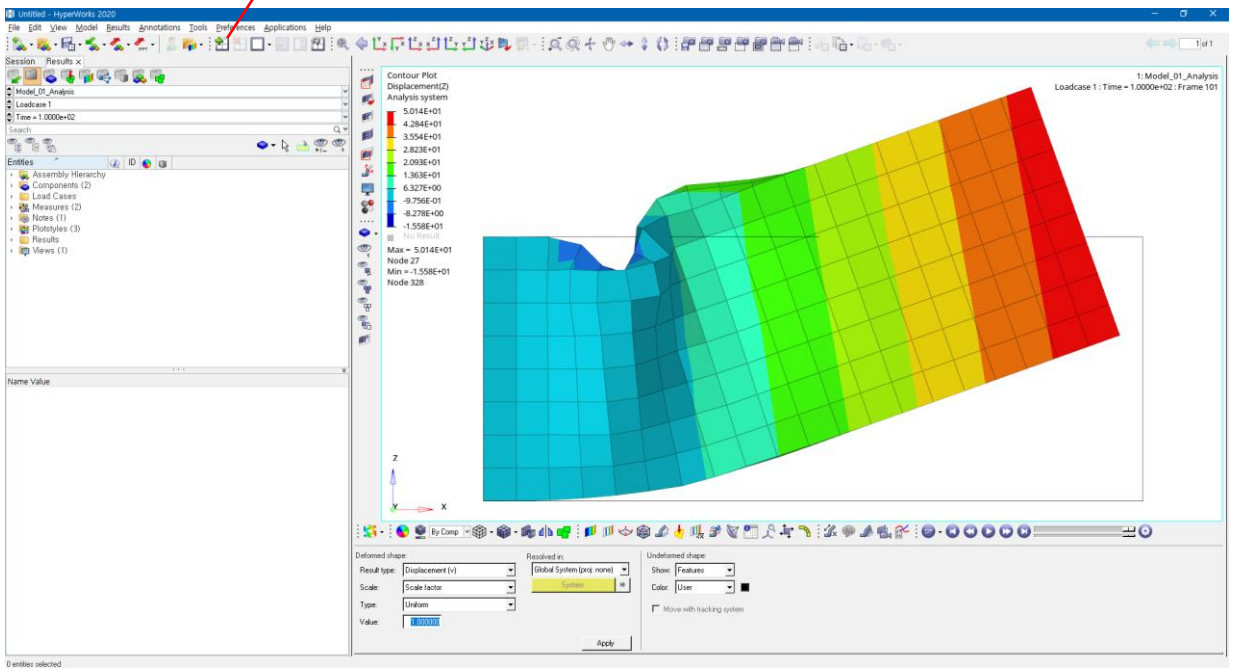
1 Radioss2020 실행

2 앞서 제작한 모델 불러온 뒤 Run 클릭





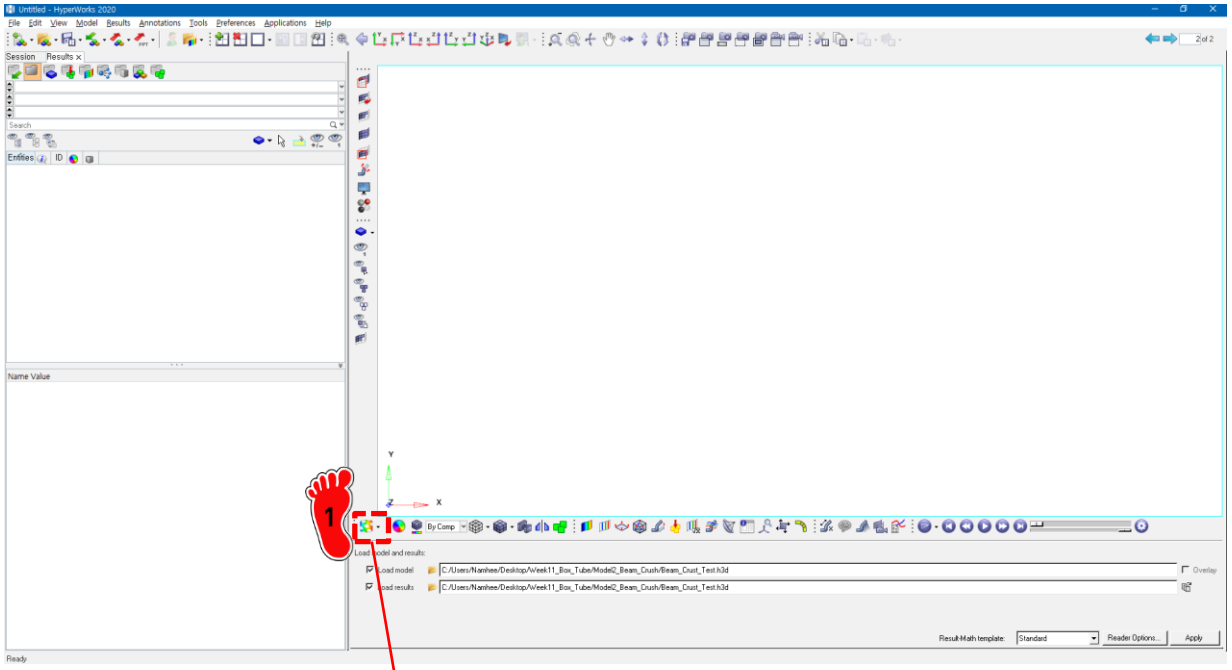
후처리 (1)

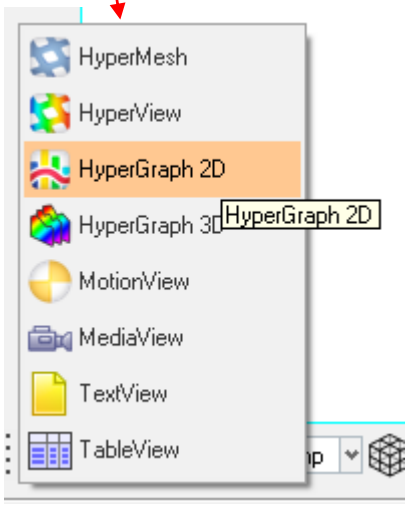


1 HyperView에서 Add Page 버튼 클릭

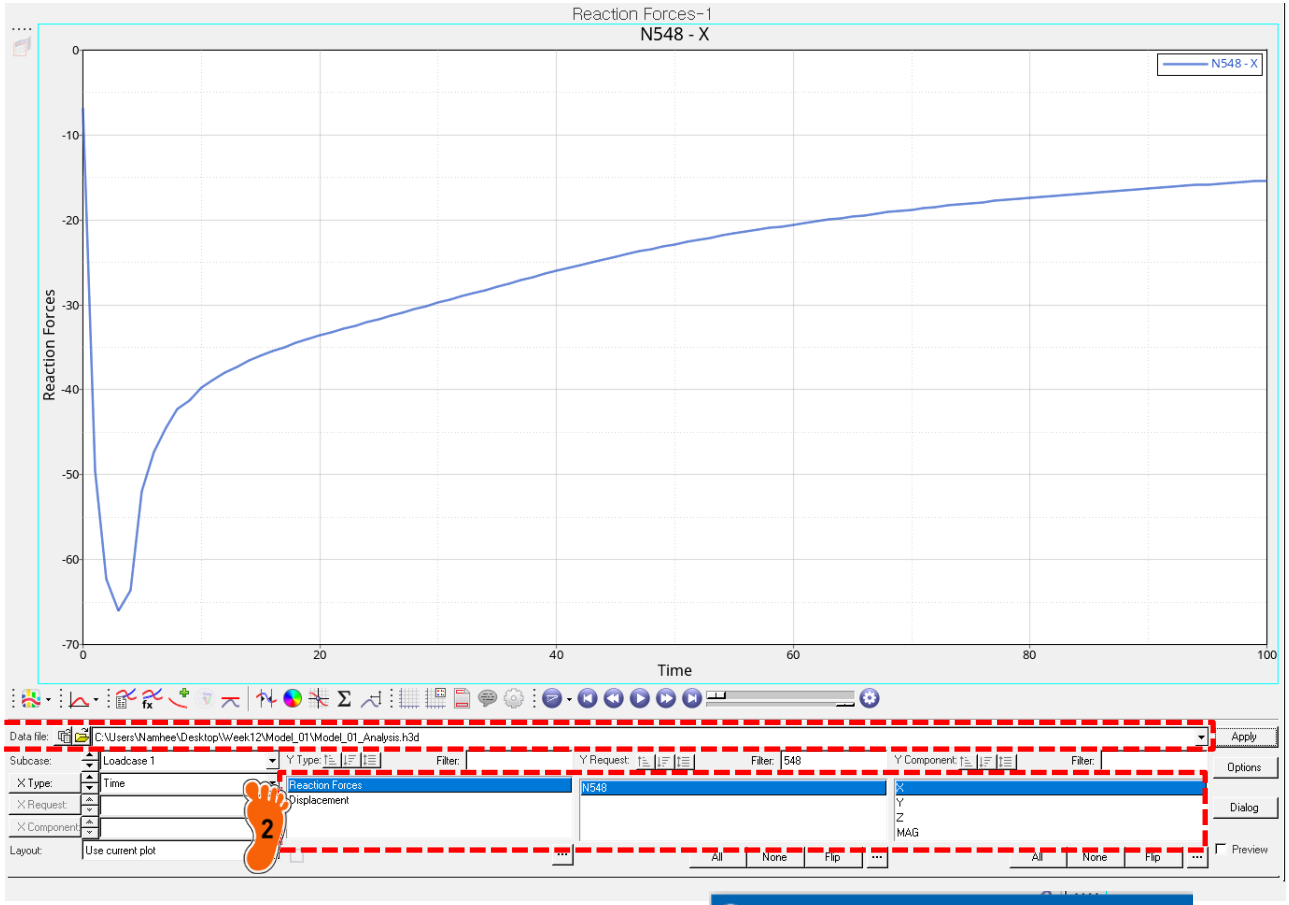
후처리 (2)

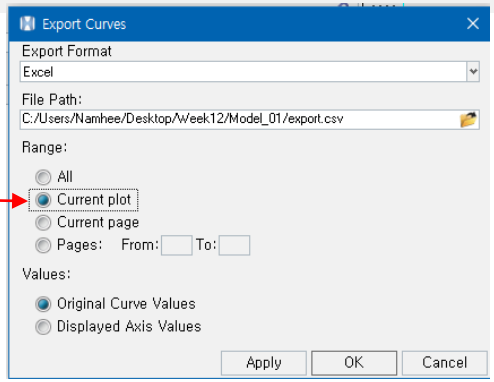
1 HyperGraph 2D 클릭





후처리 (3)



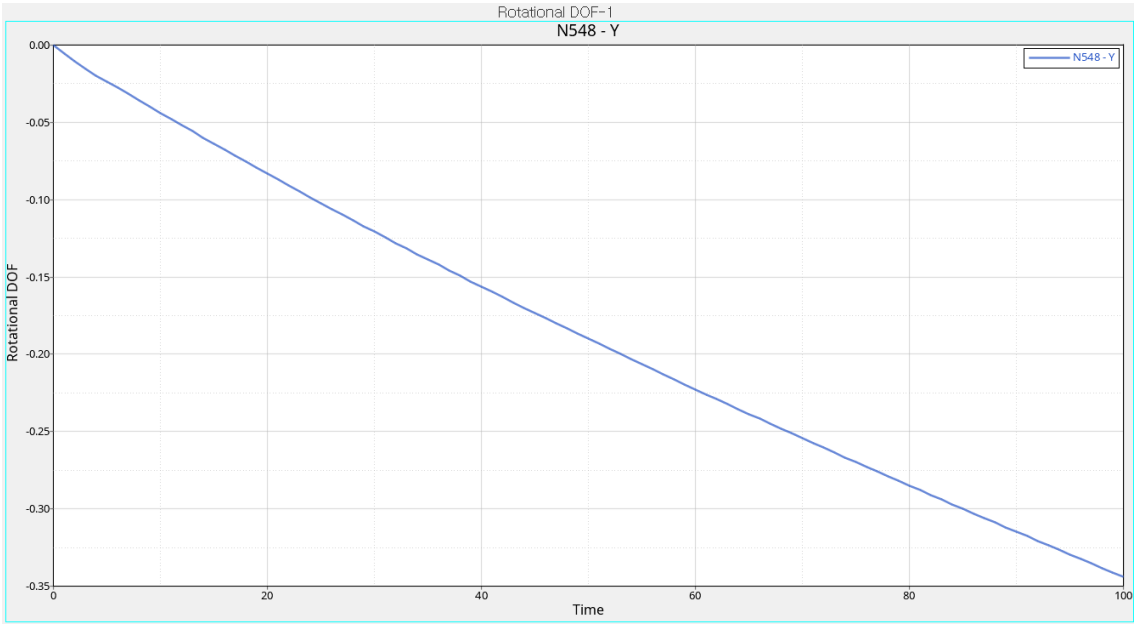


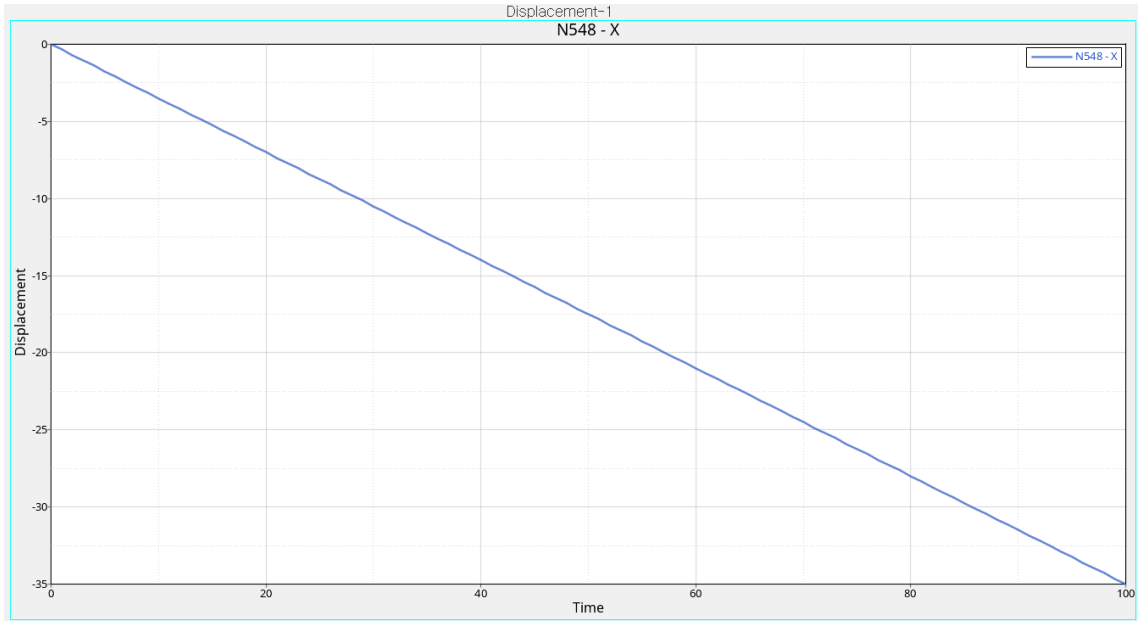
1 해석 결과 파일 불러오기 (*.h3d 파일)

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

3 그래프 csv파일로 저장

후처리 (4)



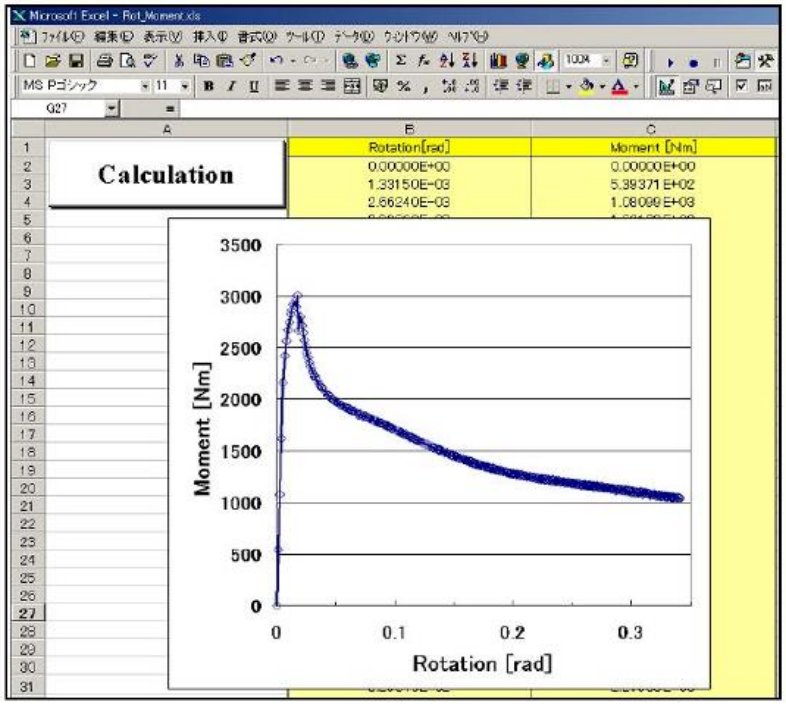


1. 새로운 그래프 추가 후
Rotational DOF 선택,
548번 절점 선택,
Y 방향으로의 그래프 확인
→ 그래프 csv파일로 저장

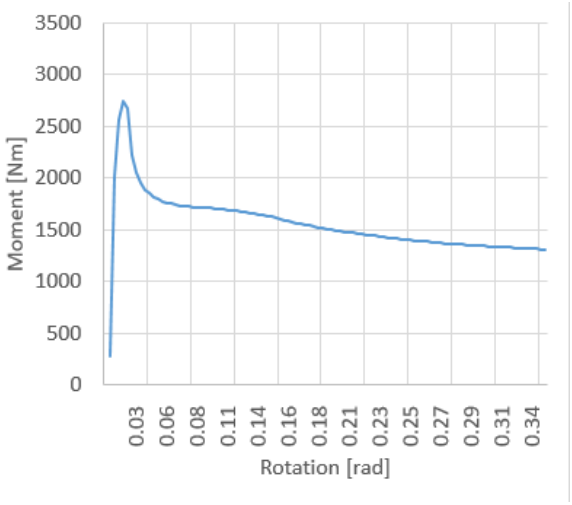
2. 새로운 그래프 추가 후
Displacement 선택,
548번 절점 선택,
X 방향으로의 그래프 확인
→ 그래프 csv파일로 저장

후처리 (5)

참고 논문 결과

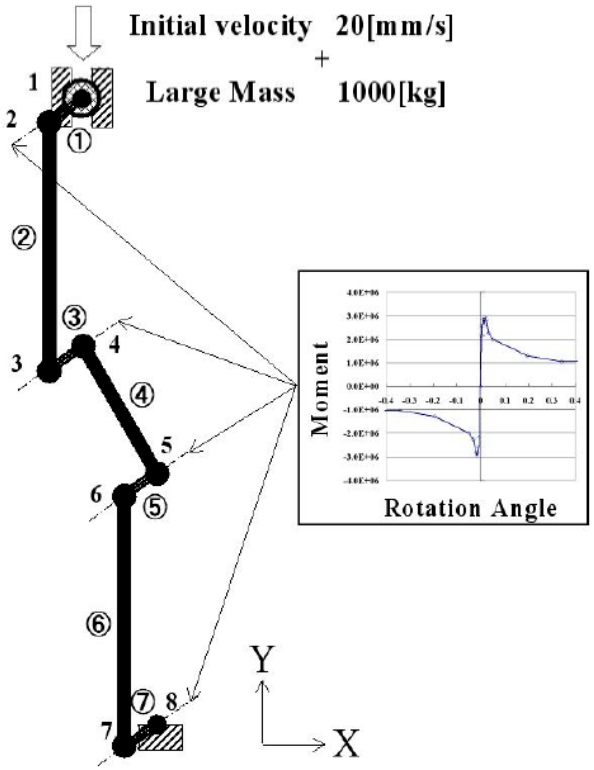


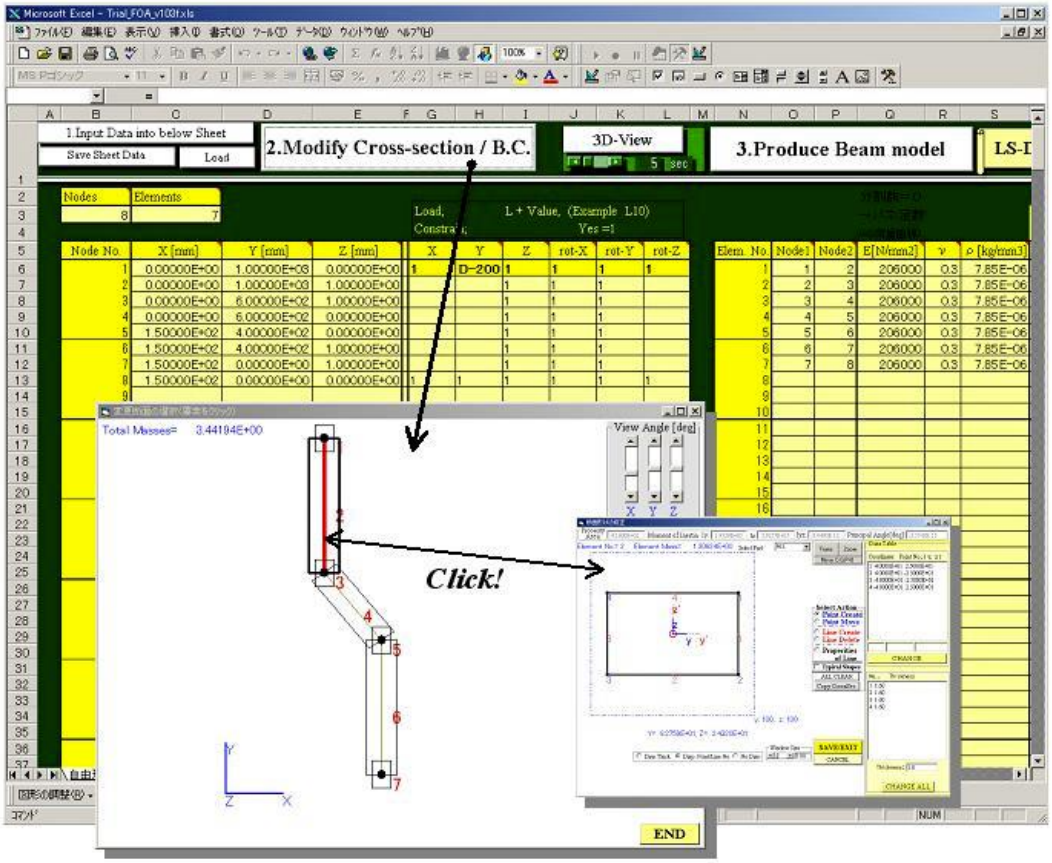
해석 결과



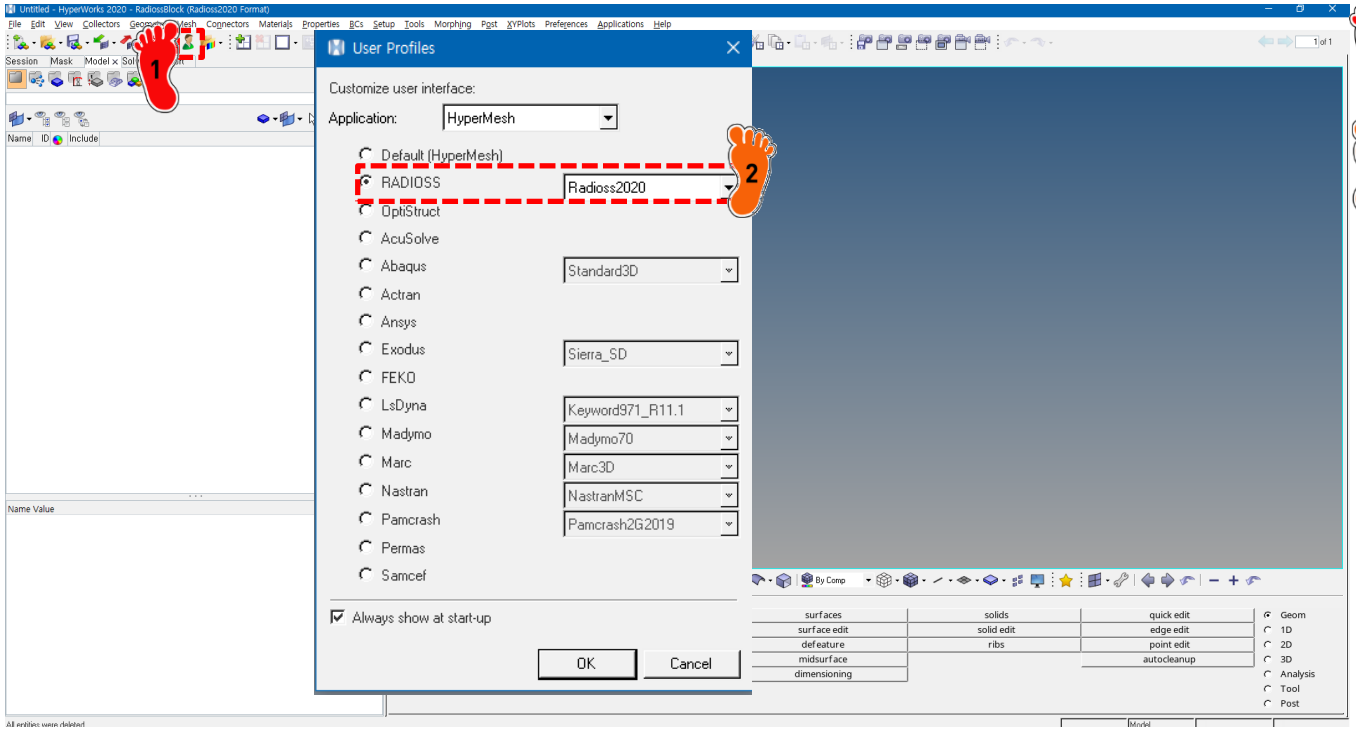
FIRST ORDER ANALYSIS
빔 요소

예제: JOINT MOMENT ANALYSIS





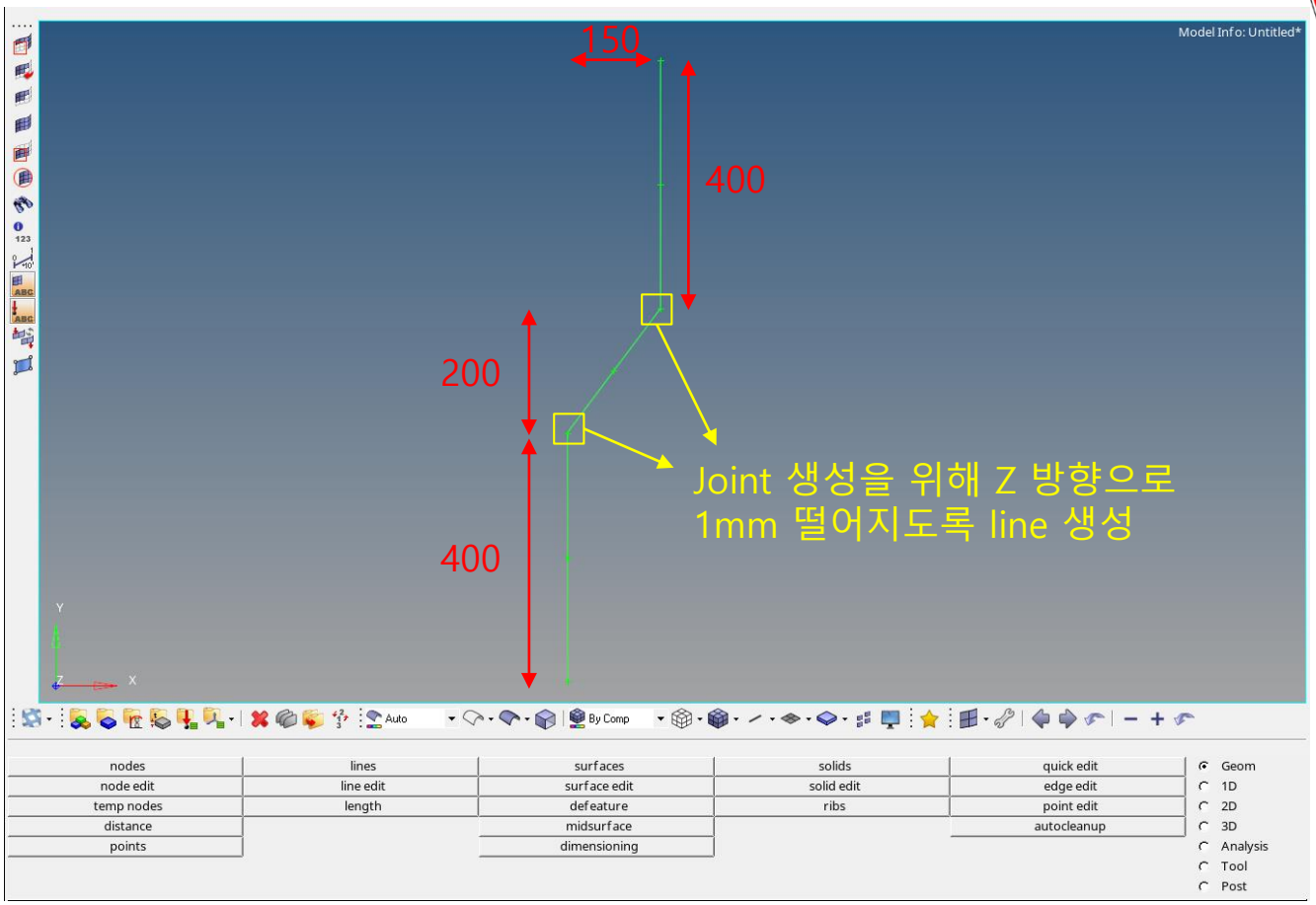
기하형상 생성 (1)



1 User Profiles ( ) 클릭

2 RADIOSS 선택

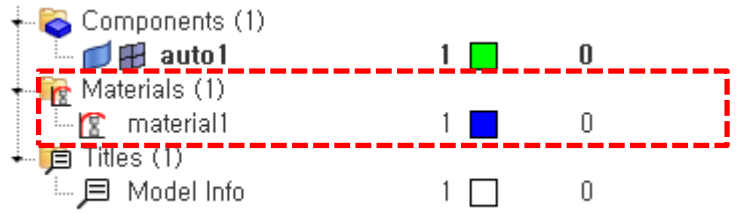
기하형상 생성 (2)



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

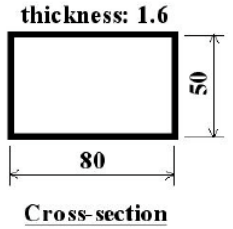
1 선형 재료 생성
Card Image: M1_ELAST
Rho_Initial: 7.85e-9
E: 206 GPa
Nu: 0.3

2 1차원 특성 생성
(사각 단면 구조 적용)

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

1 Beam section 생성



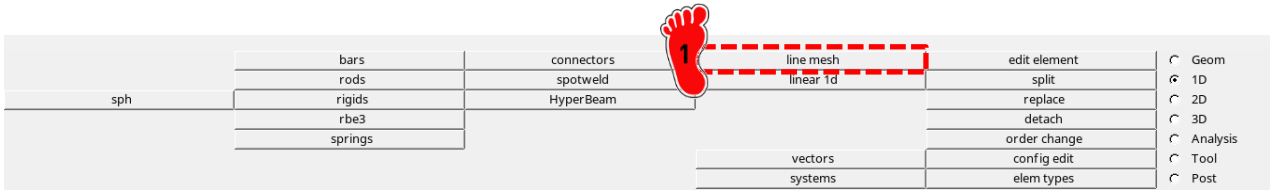
Name	Value
Solver Keyword	/PROP/BEAM/
ID	1
Name	property1
Color	
Include	[Master Model]
Defined	☑
User Comments	Hide In Menu/Export
Card Image	P3_BEAM
⊞ Hyperbeamsection	(1) beamsection1
Regular_OR_encrypted_flag	Regular

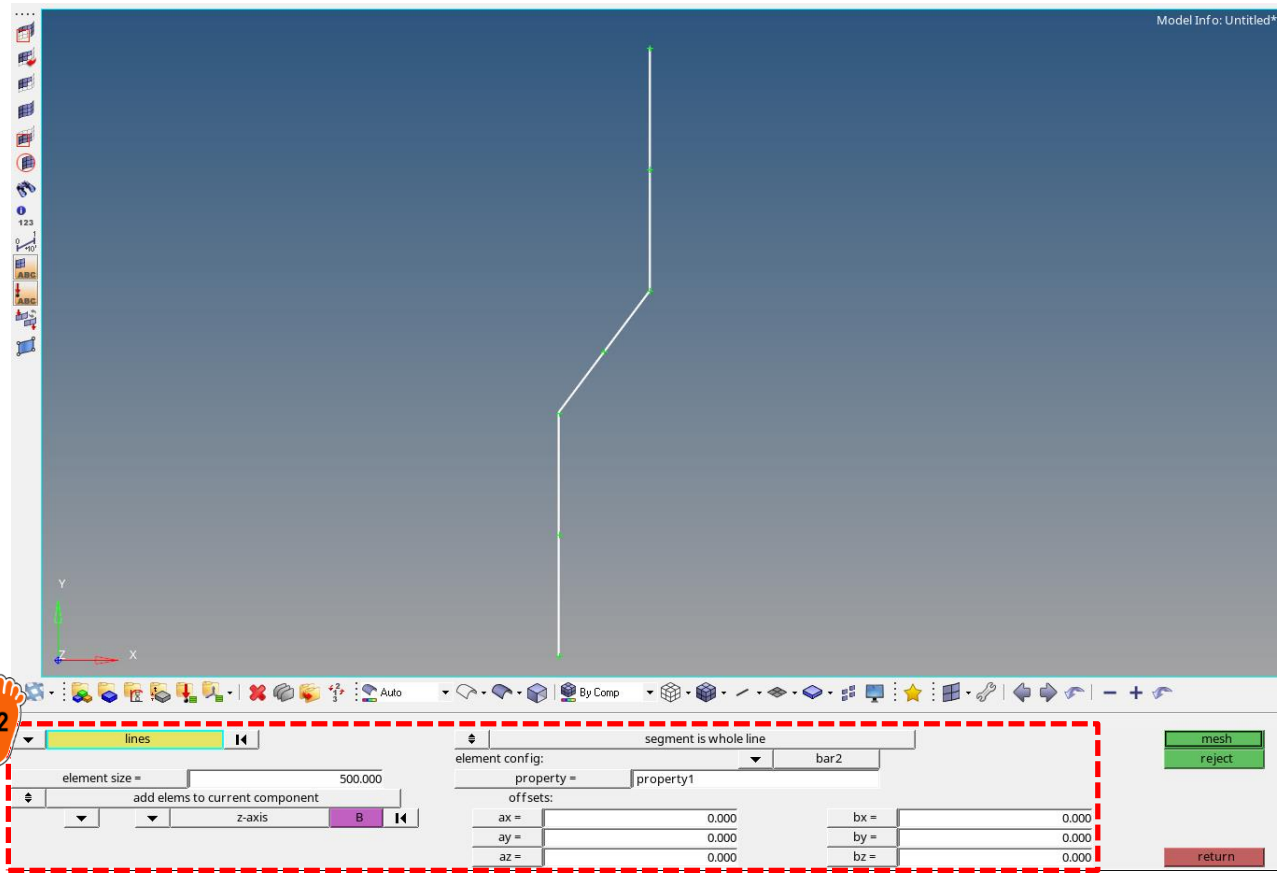
2 Property 생성
Card Image: P3_BEAM
Beam section 설정

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

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

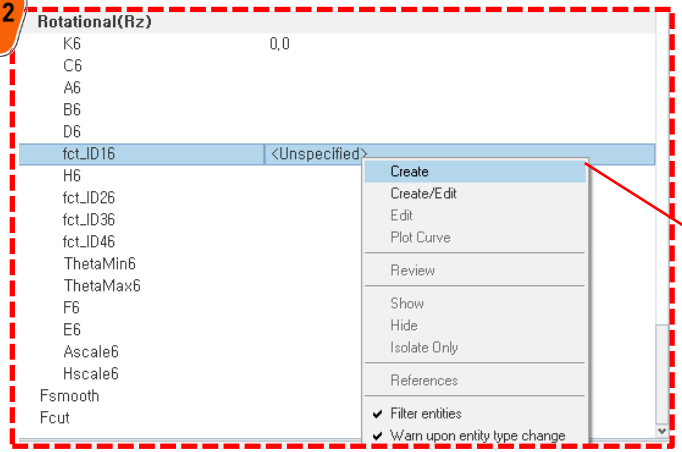
1 Joint property 생성
Card Image: P8_SPR_GENE

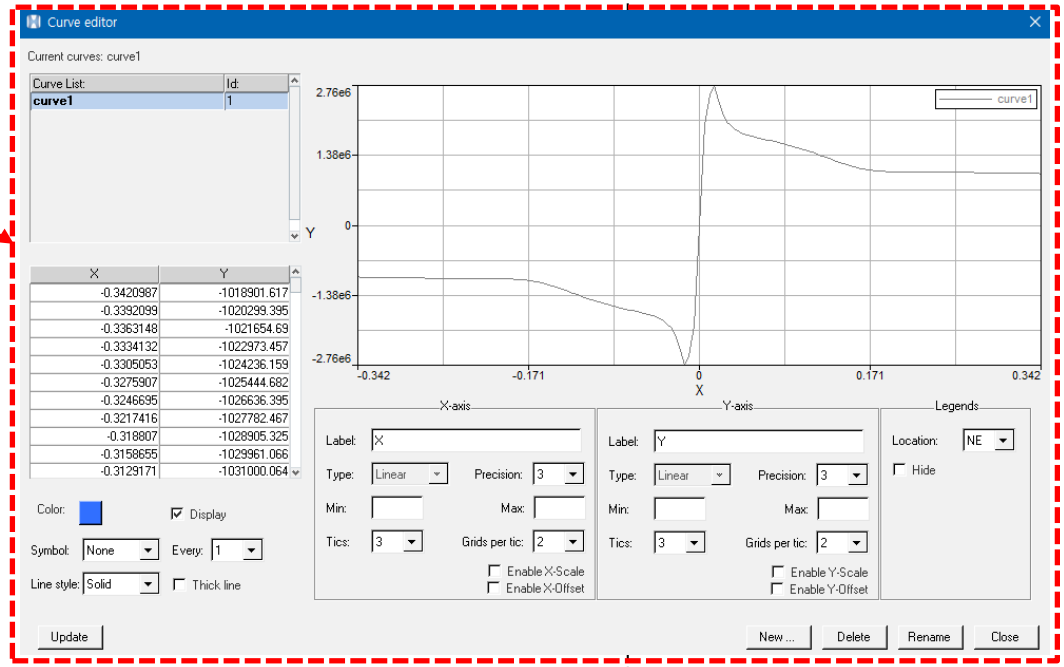
Mass: 0.0001
Inertia: 0.0001

Tx,Ty,Tz,Rx,Ry: K1 = 1e10

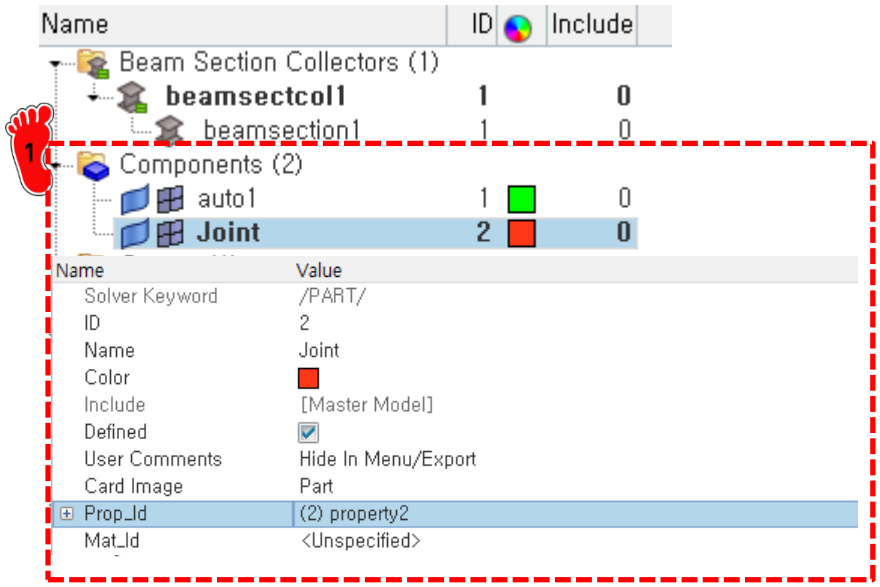
2 Rz: K1 = 0,
fct_ID16 활성화하여 회전 강성 테이블 입력

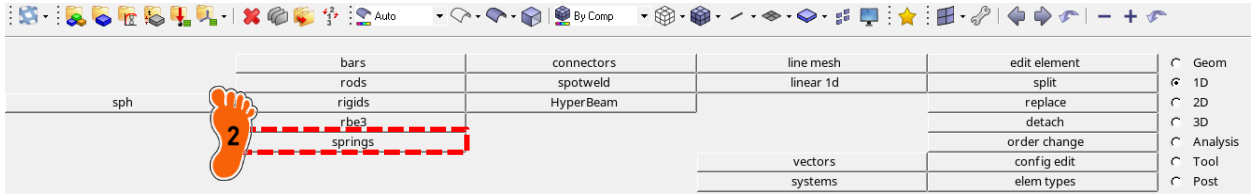
2

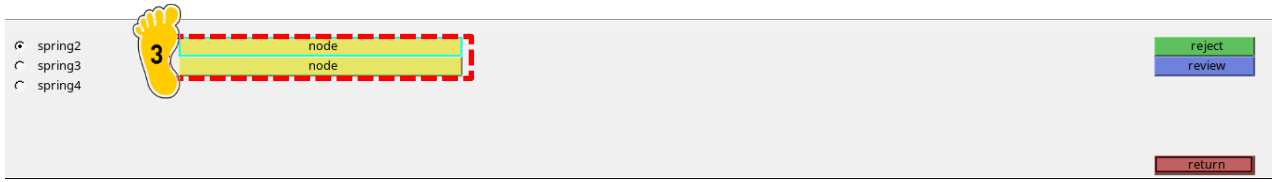




조인트 요소 생성 (2)

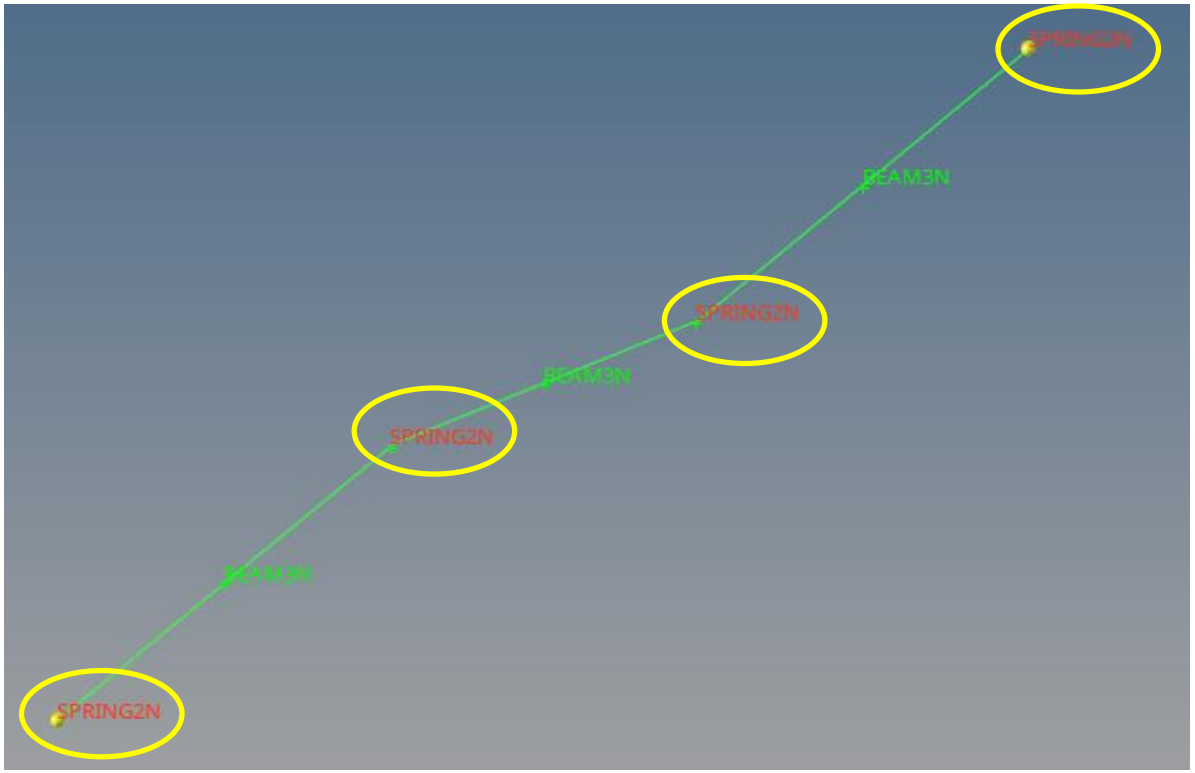






1. Joint component 생성 후 property 부여
2. 1D>spring
3. spring2 생성에서 절점 두 개 선택하여 joint 요소 생성

조인트 요소 생성 (3)

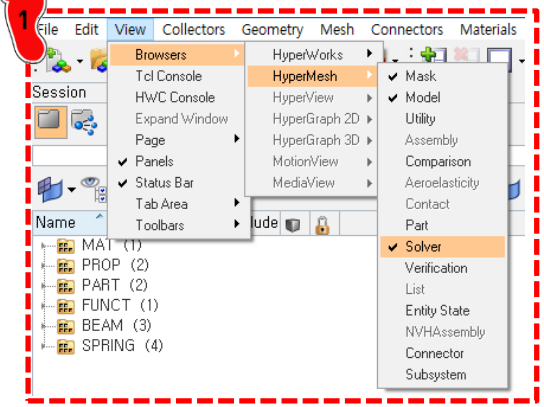


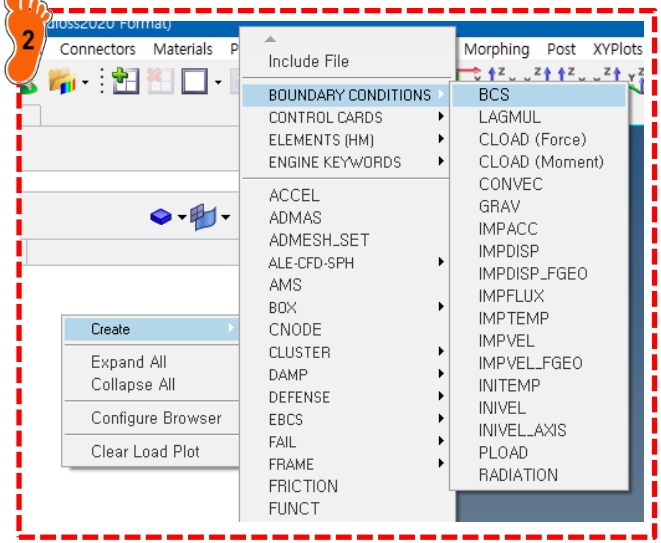
1 네 군데 joint 요소 생성

2 양 끝단에는 temp node를 생성하여 joint를 생성함

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

구속조건 설정 (1)



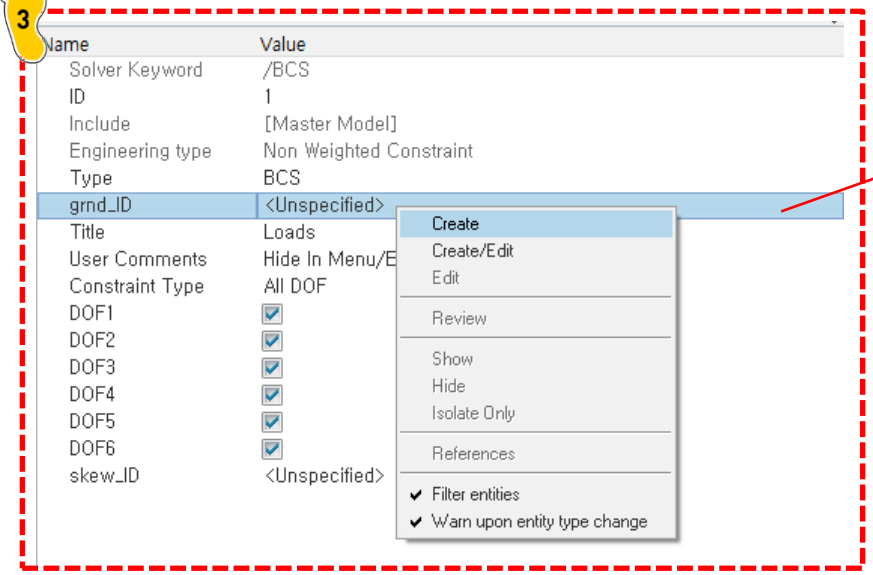


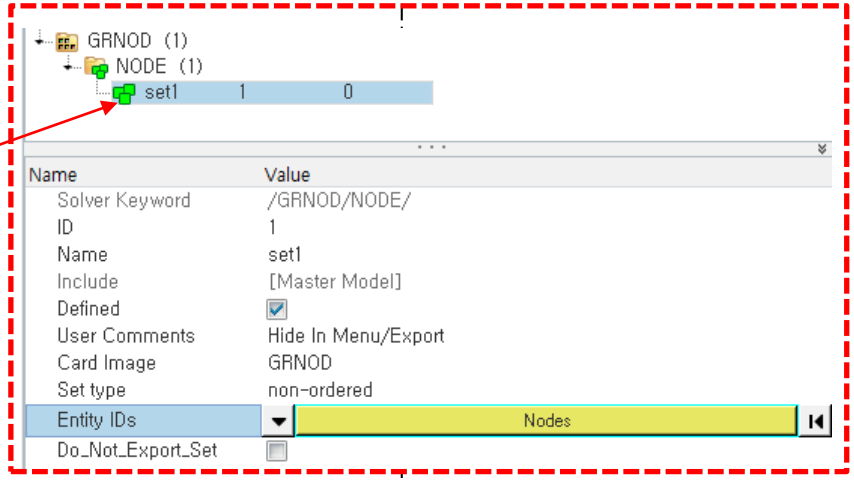
1. View>Browsers>HyperMesh>Solver 클릭

2. Solver 탭에서 마우스 우클릭 하여 BCS 생성

3. grnd_ID 우클릭 하여 절점 set 생성
→ 고정 구속할 노드 추가

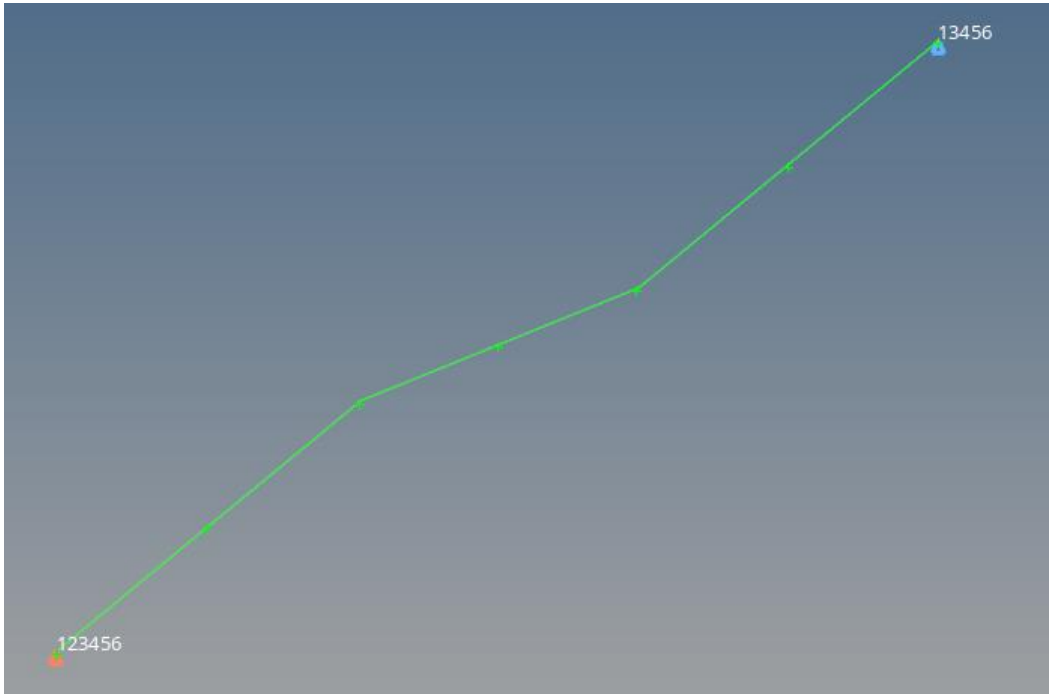
DOF 1~6 모두 구속





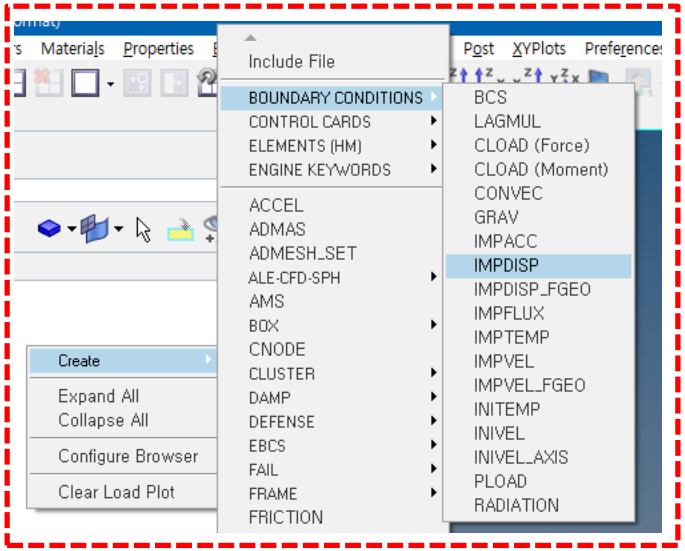
구속조건 설정 (2)

Name	Value
Solver Keyword	/BCS
ID	2
Include	[Master Model]
Engineering type	Non Weighted Constraint
Type	BCS
grnd_ID	<Unspecified>
Title	SPC
User Comments	Hide In Menu/Export
Constraint Type	All DOF
DOF1	☑
DOF2	☐
DOF3	☑
DOF4	☑
DOF5	☑
DOF6	☑



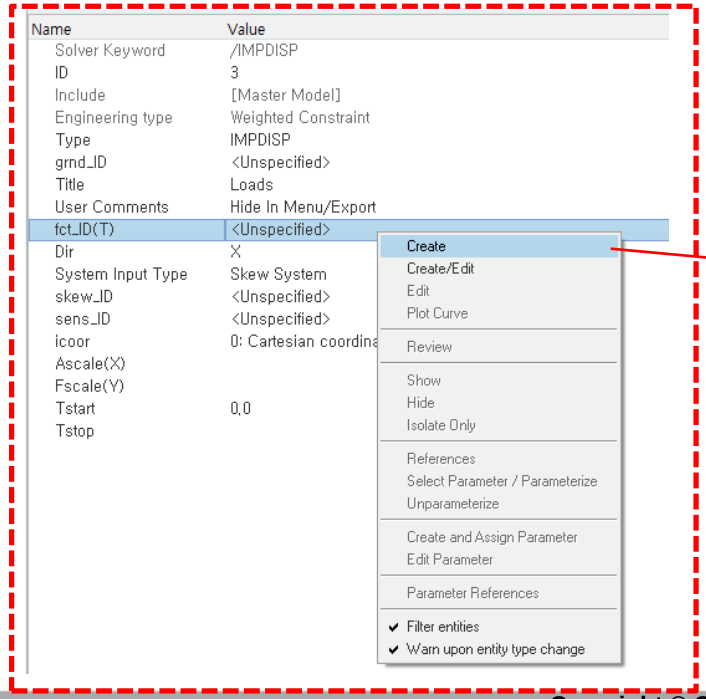
1 같은 방법으로 반대편 절점은 DOF2 제외하고 모두 구속

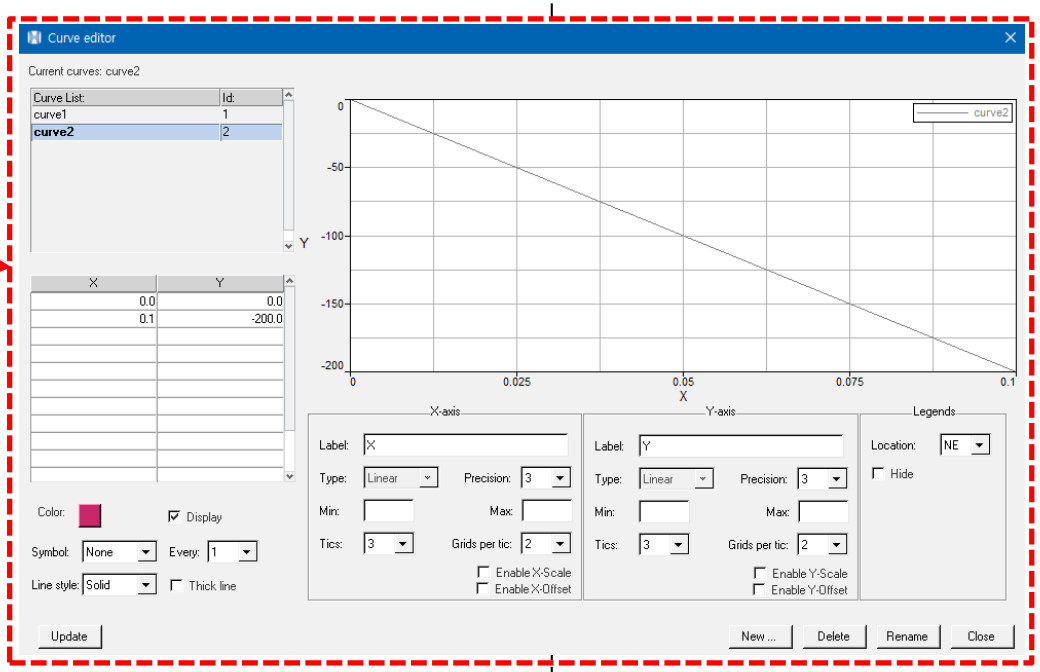
변위 조건 설정 (1)



1 Solver탭 우클릭하여 IMPDISP 생성

2 fct_ID(T) 우클릭하여 시간-변위 함수 생성





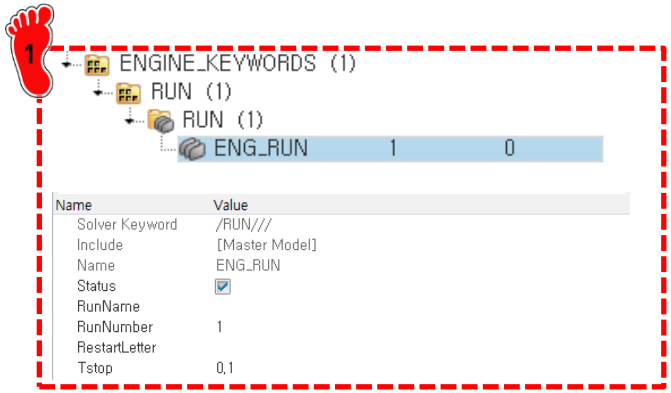
변위 조건 설정 (2)

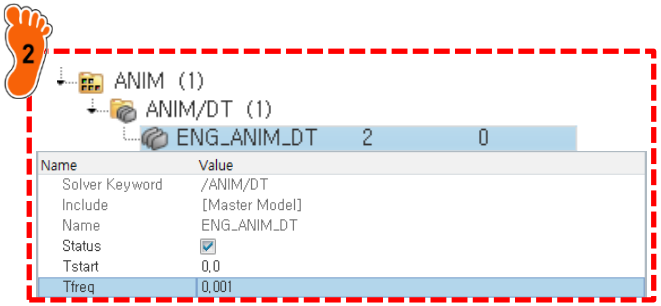
Name	Value
Solver Keyword	/IMPDISP
ID	3
Include	[Master Model]
Engineering type	Weighted Constraint
Type	IMPDISP
grnd_ID	<Unspecified>
Title	Loads
User Comments	Hide In Menu/Export
fct_ID(T)	(2) curve2
Dir	Y
System Input Type	Skew System
skew_ID	<Unspecified>
sens_ID	<Unspecified>
icoor	0: Cartesian coordinates
Ascale(X)	
Fscale(Y)	
Tstart	0.0
Tstop	

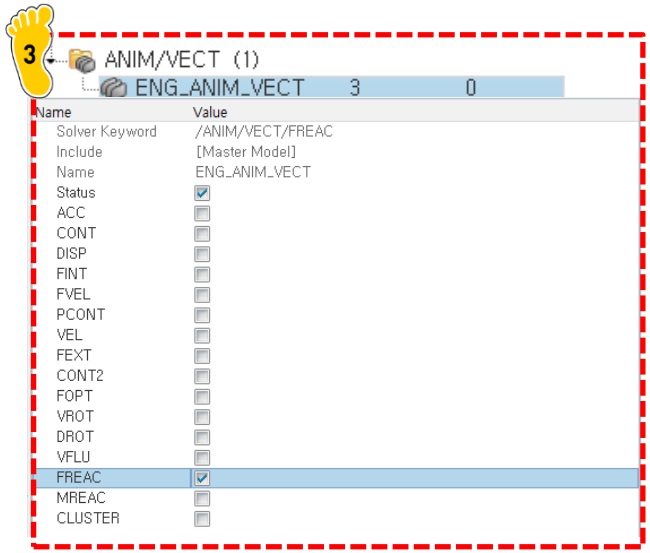
1. grnd_ID 클릭하여 set 생성 후 절점 추가

2. Dir은 Y로 설정

해석 케이스 설정





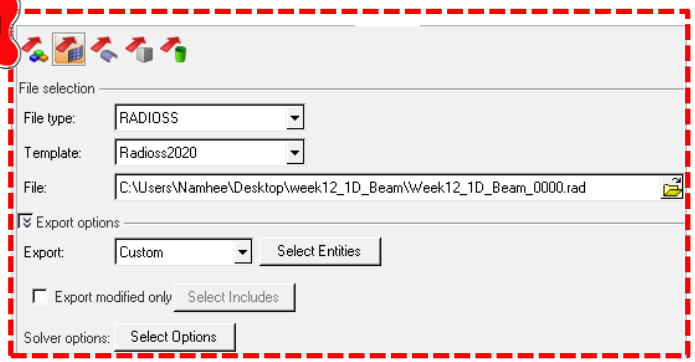


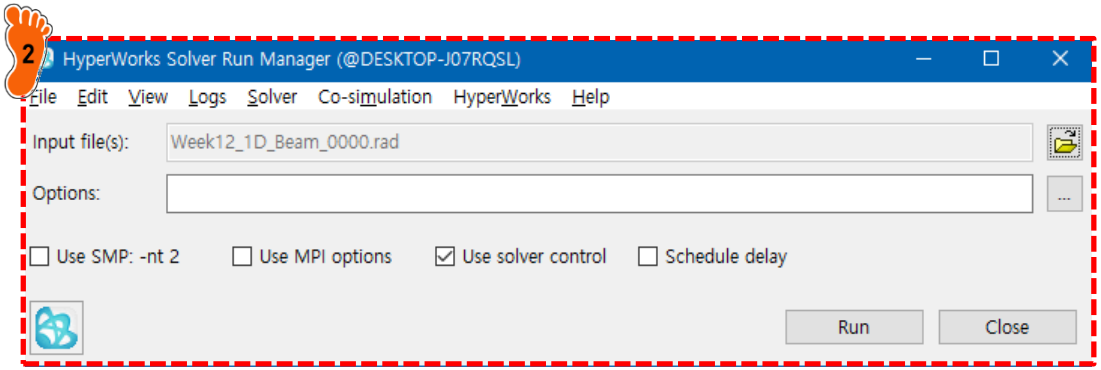
1 [ENG_RUN]
RunMumber: 1
Tstop: 0.1

2 [ENG_ANIM_DT]
Treq: 0.001

3 [ENG_ANIM_VECT]
FREAC

해석 실행



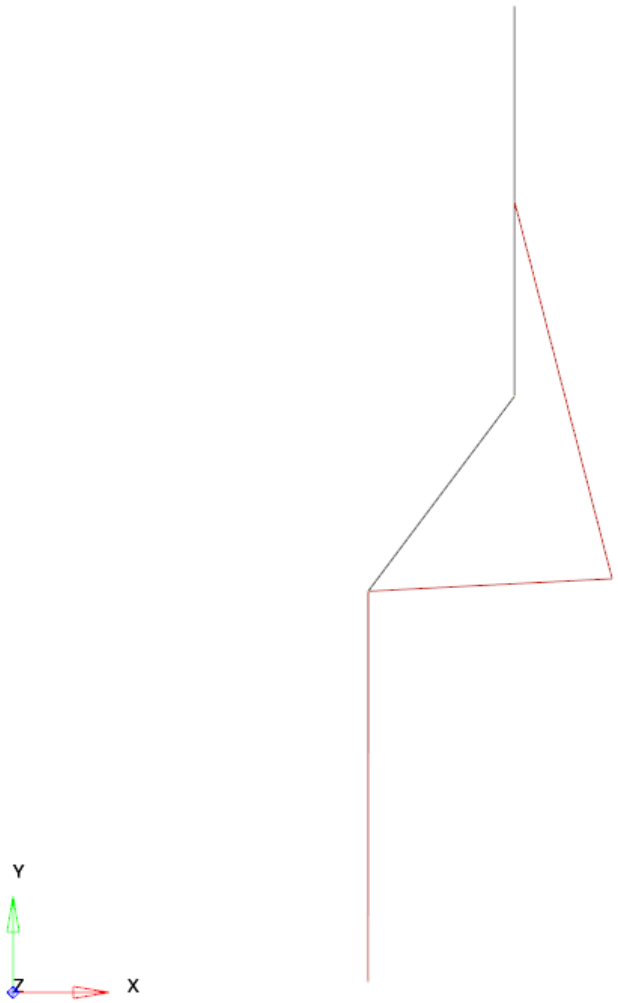


1 모델 export

2 Radioss에서 해석 실행

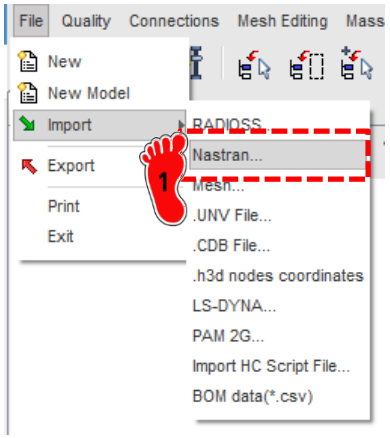
후처리

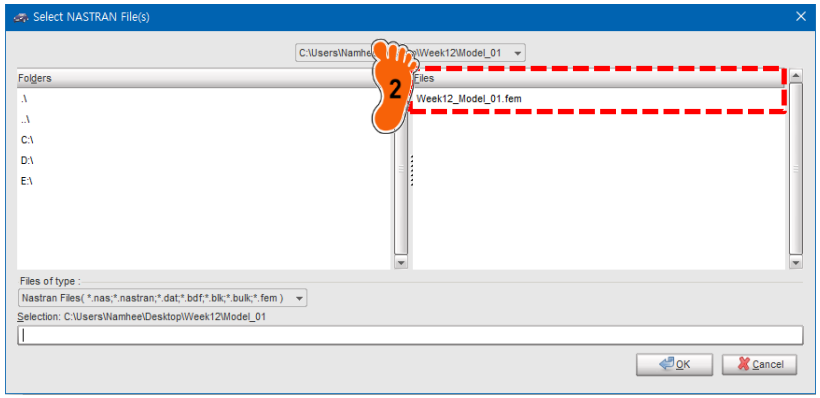
1 변형된 형상 확인

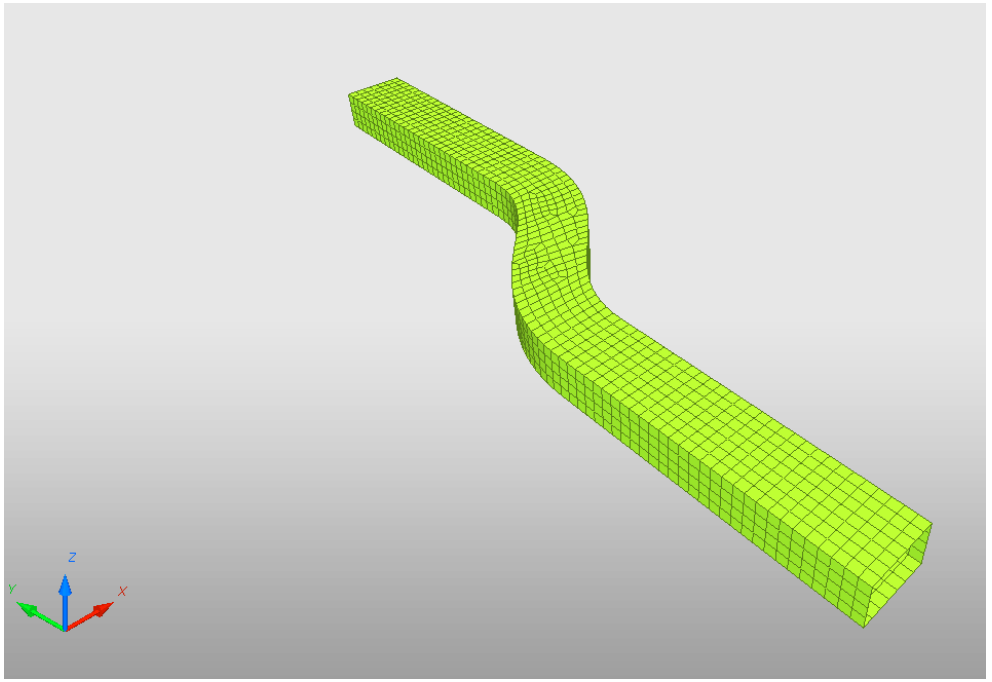


MID RAIL ANALYSIS
쉘 요소

모델 불러오기



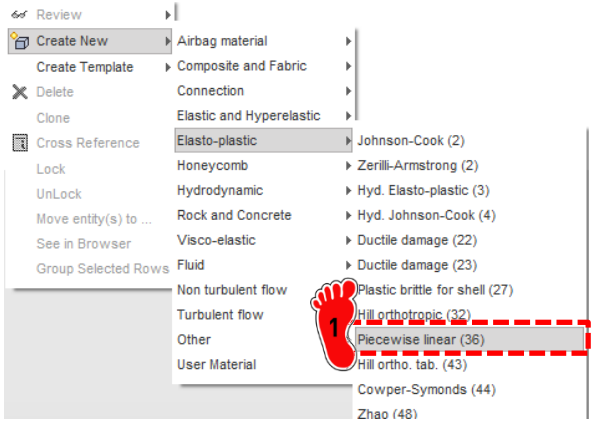


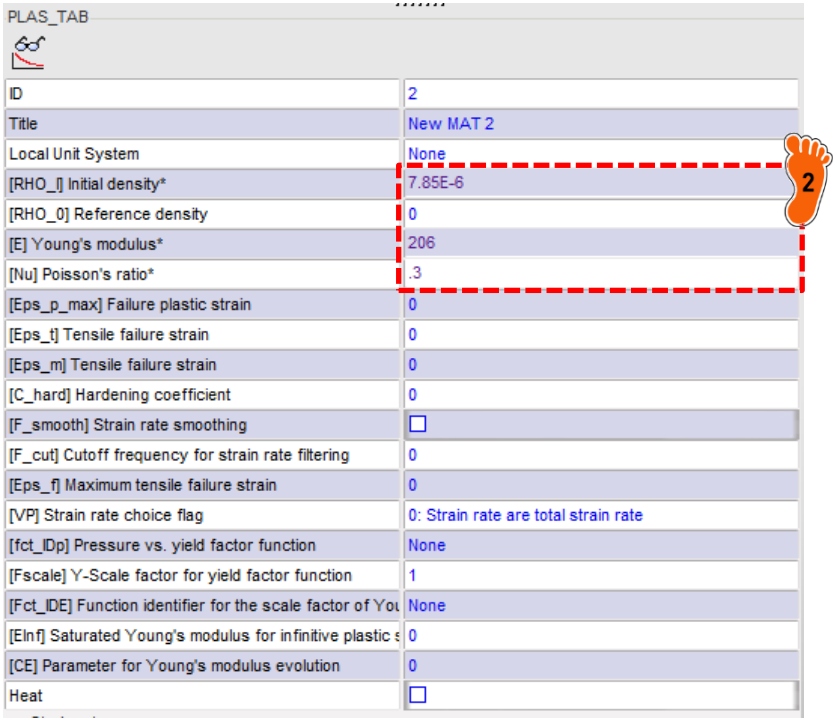


1 File > Import > Nastran 클릭

2 모델 파일 클릭 후 불러오기 (Week12_Model_02.fem)

재료 물성 입력 (1)

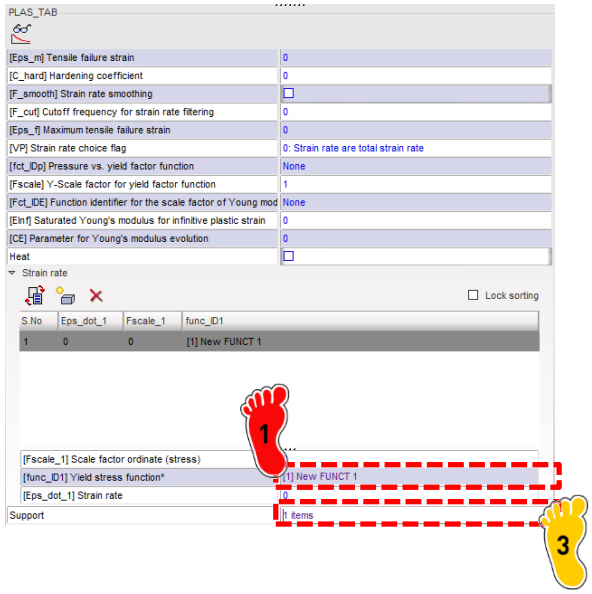


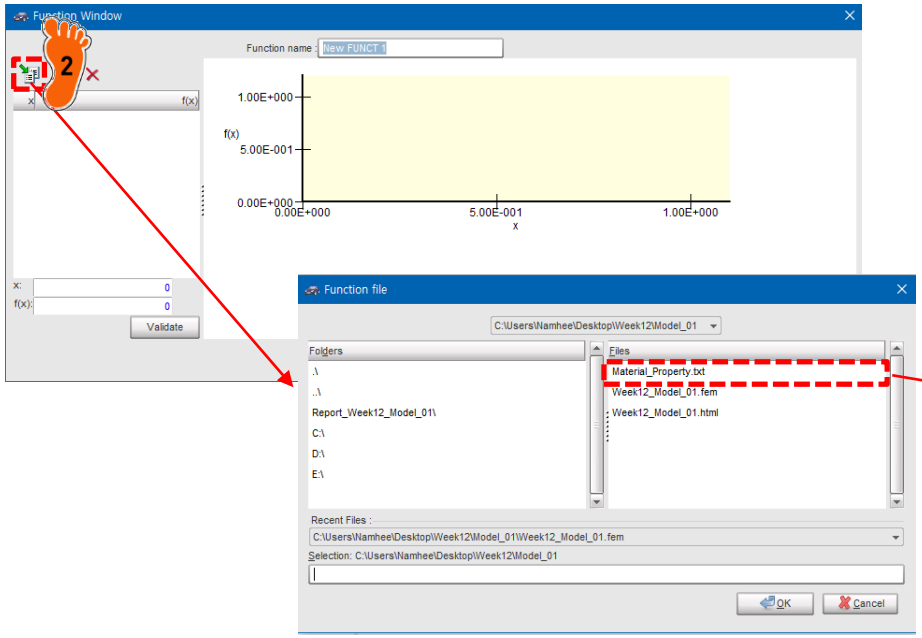


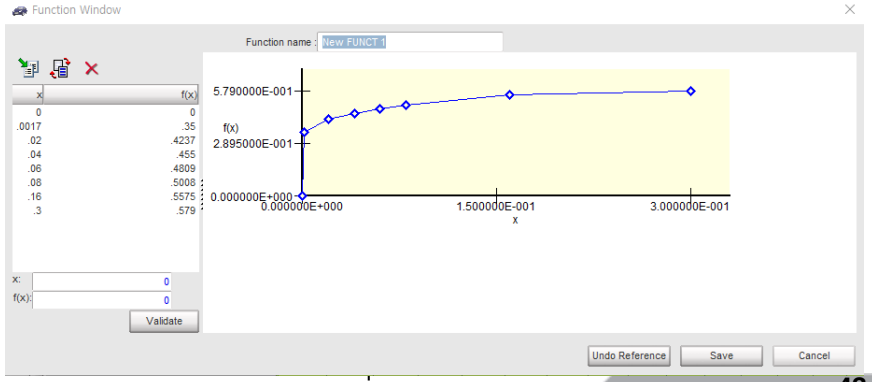
1 Piecewise linear(36) 재료 생성

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

재료 물성 입력 (2)

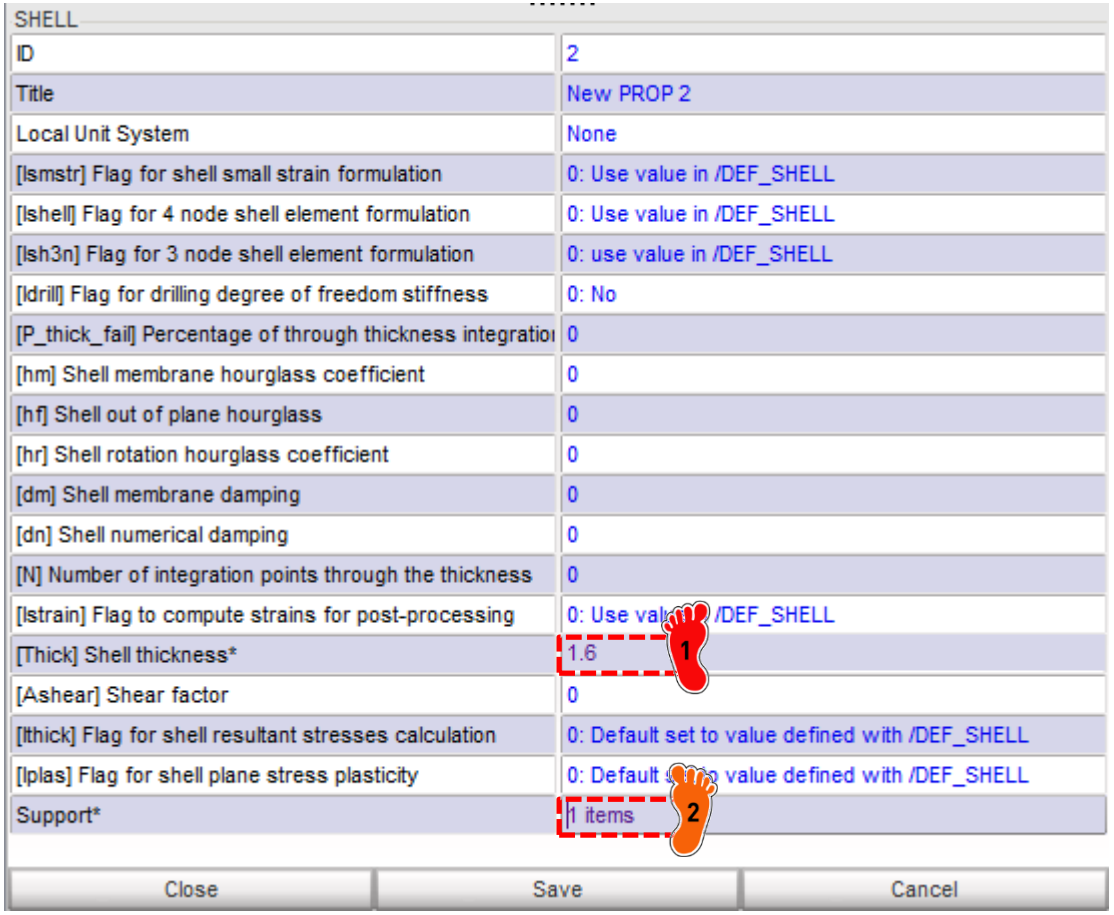






1. Stress-Strain curve 입력 (마우스 우클릭 후 "Define function" 클릭)
2. "Import curve" 클릭 후 "Material Property.txt" 파일 불러오기
3. 모델에 재료 적용

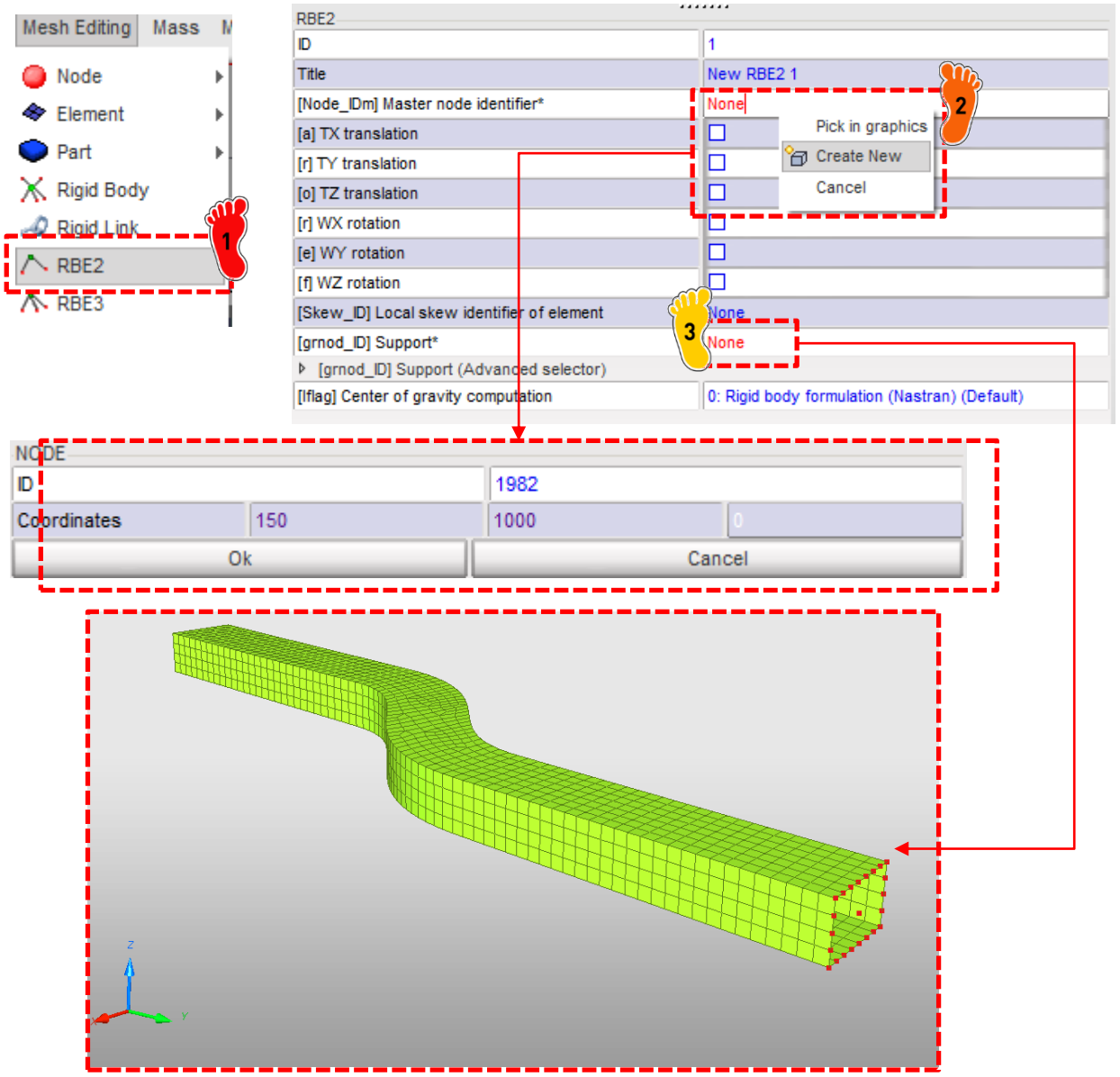
특성 입력



1. 쉘 property 생성 후 두께 1.6mm 입력

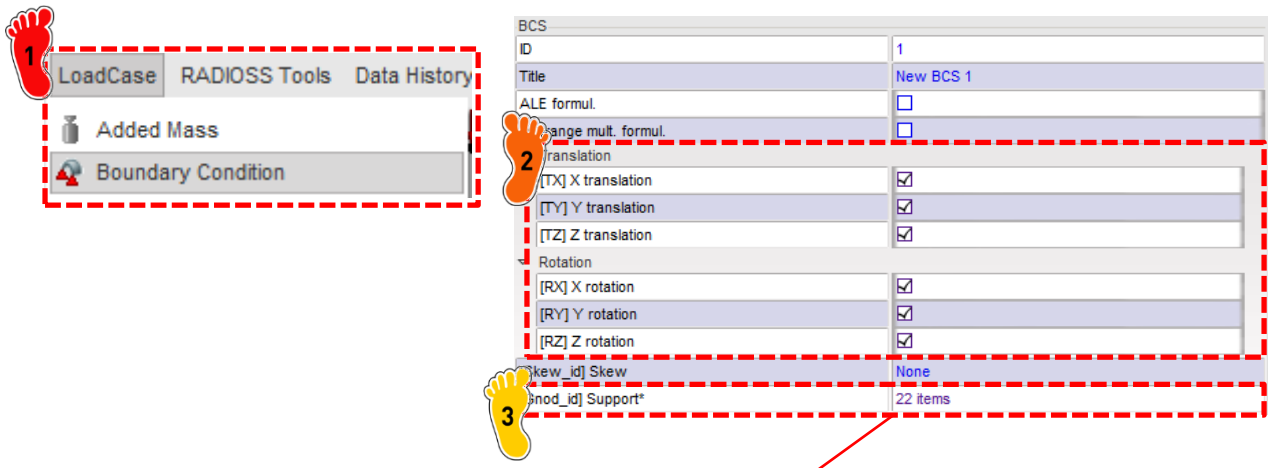
2. 모델에 특성 적용

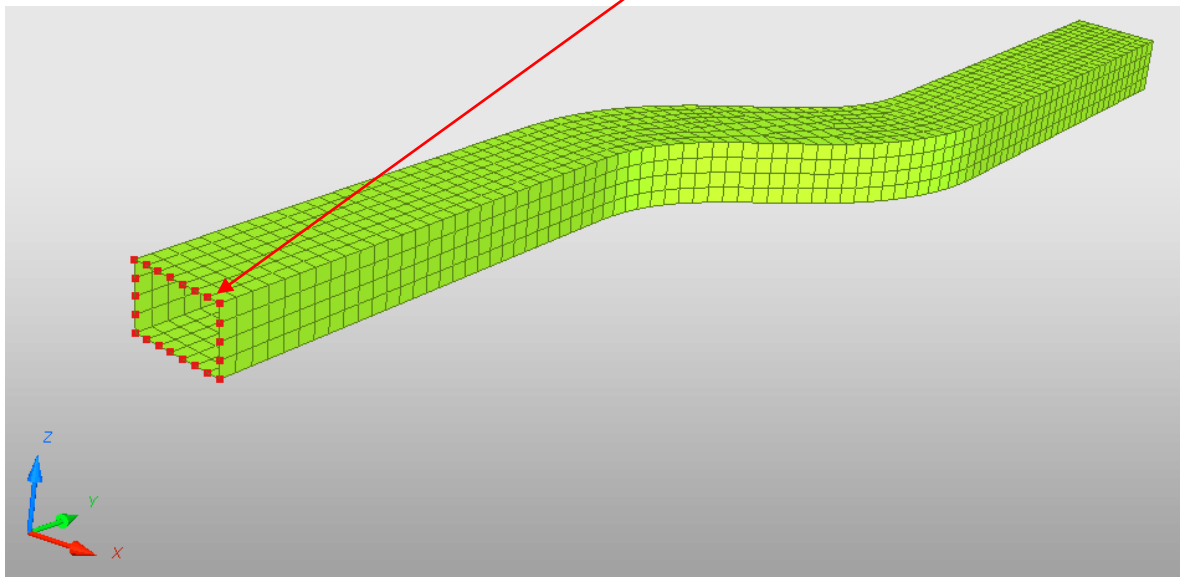
RBE2 생성



1. RBE2 생성
2. Master node 좌표: (150, 1000, 0)
3. Support: Y방향 끝단 노드 선택

구속조건 설정 (1)



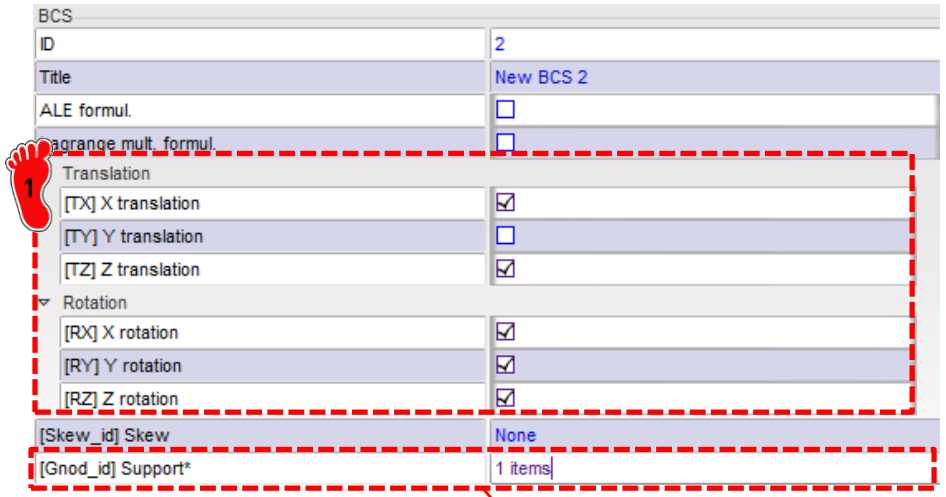


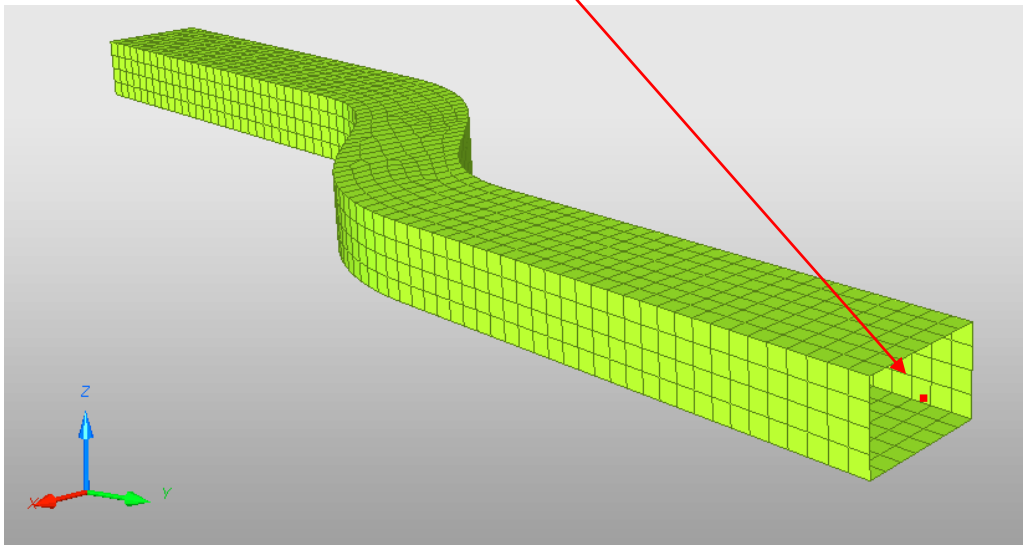
1 Boundary condition 생성

2 완전 구속 설정

3 구속할 절점 선택

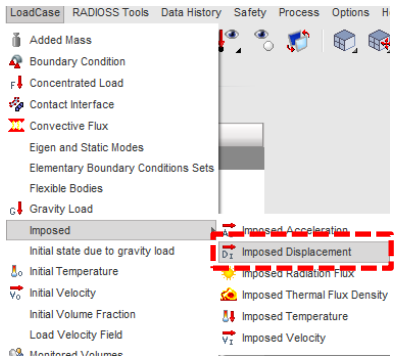
구속조건 설정 (2)

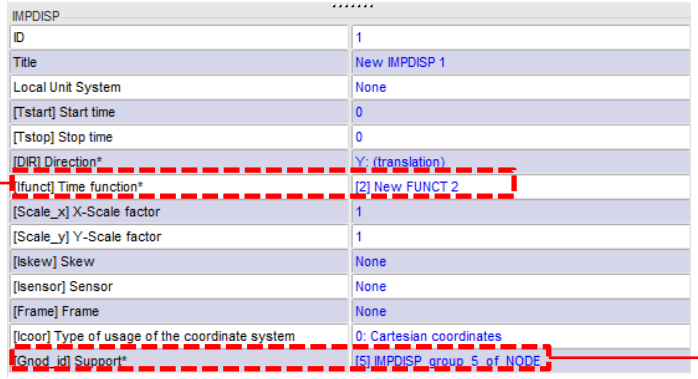


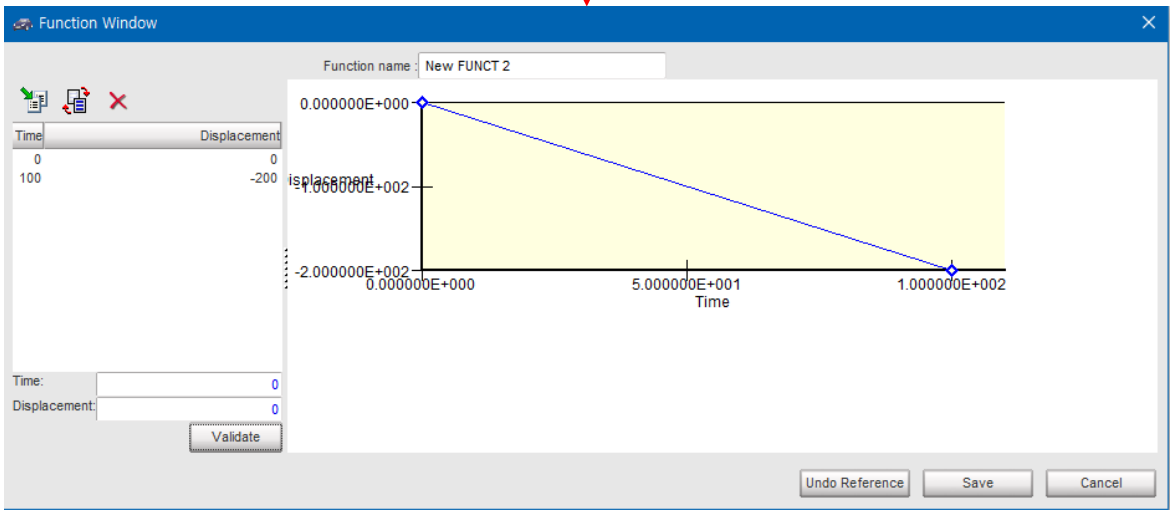


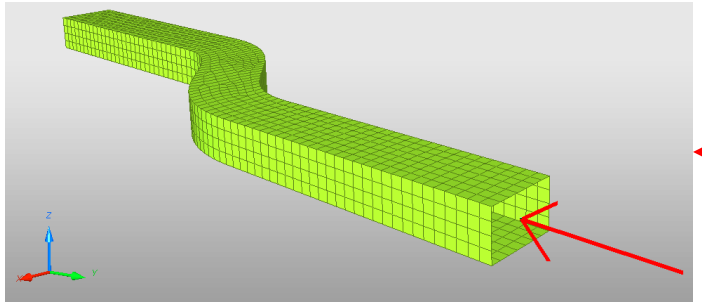
강체 중심 절점은 Y 방향 구속조건을 제외한 모든 자유도 구속
(Tx,Tz,Rx,Ry,Rz 구속)

변위조건 설정



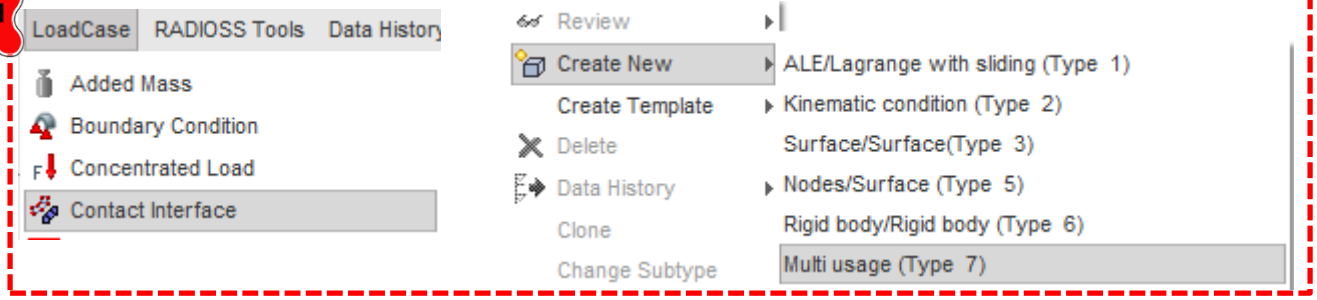






1. Imposed displacement 생성
2. 시간-변위 그래프 생성 (0,0), (100,-200)
3. Support: 앞서 만든 RBE2 요소의 master node 선택

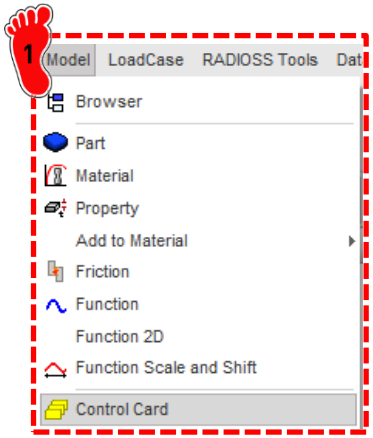
접촉조건 설정

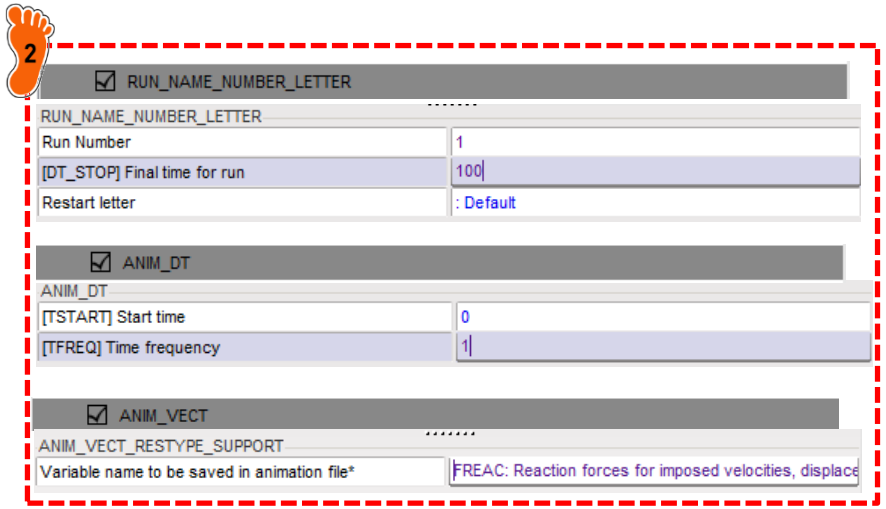


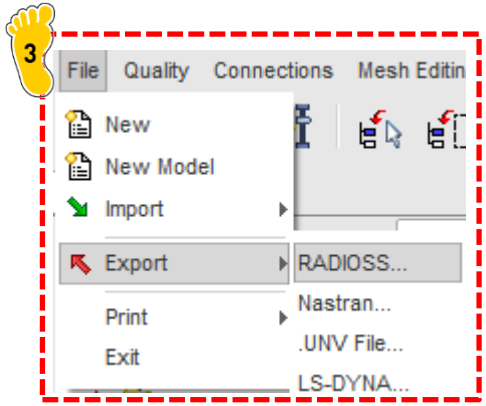
1 LoadCase > Contact Interface클릭,
마우스 우클릭 후 "Multi usage (Type 7) 클릭

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

해석 케이스 설정







1 Model > Control Card클릭

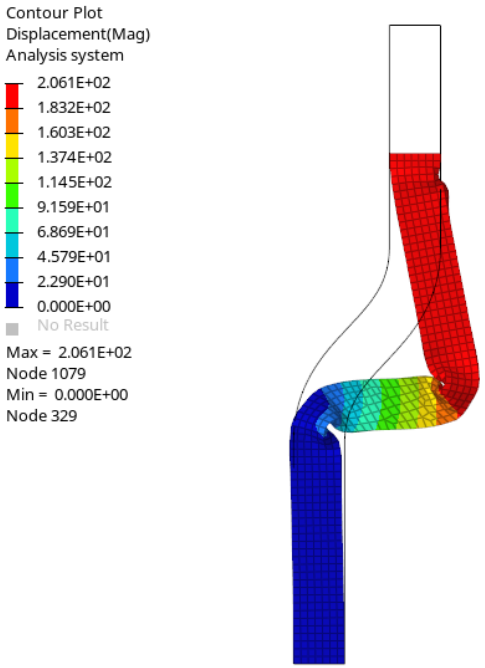
2 파라미터 설정

3 모델 export (RADIOSS 선택)

후처리 (1)

참고 논문 결과

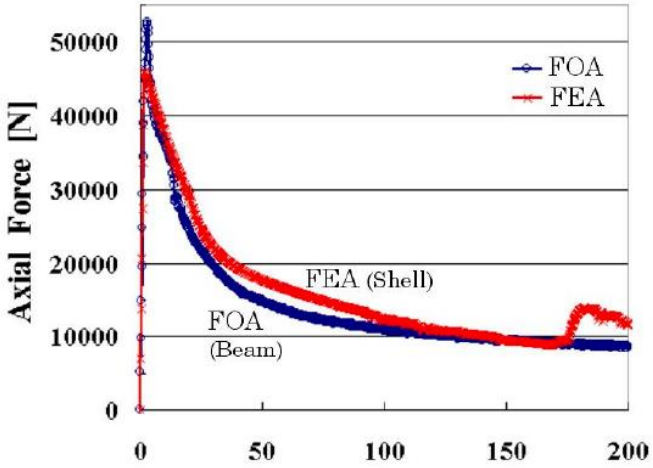
해석 결과



후처리 (2)

1 강체 중앙 절점의 Y 방향 반력 그래프 표시

참고 논문 결과



해석 결과