

SubStr

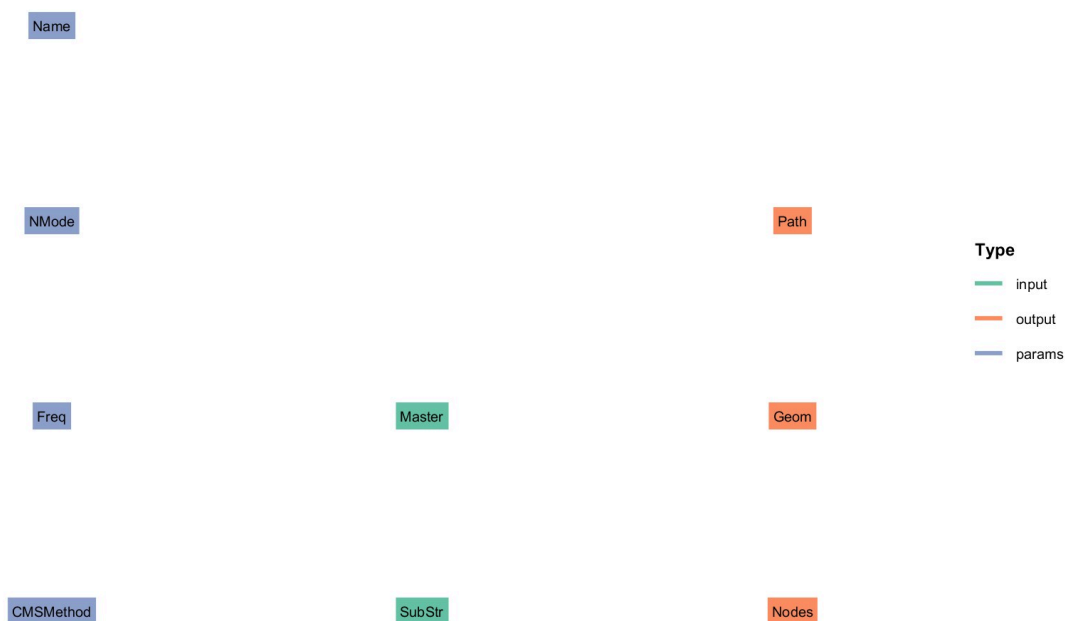
Xie Yu

1 介绍

SubStr类用于生成子结构(在ANSYS中称为子结构或者超单元，在simpack中称为柔体)，它可以导出ANSYS的.sup文件和simpack的.fbi文件。子结构就是将一组单元用矩阵凝聚为一个单元的过程。这个单一的矩阵单元称为超单元。在ANSYS分析中，超单元可以象其他单元类型一样使用。

2 类结构

Object Structure



输入 input:

- Master: 主节点
- SubStr : 子结构的网格模型

参数 params:

- Name : 名称
- NMode : 模态数量，默认数量50
- Freq: 频率范围，默认0~2000Hz
- CMSMehod: CMS方法

参照ANSYS帮助文件，主要有以下三种方法，默认为‘FIX’。

CMSMethod	Description
FIX	Fixed-interface method.
FREE	Free-interface method.
RFFB	Residual-flexible free-interface method

输出 output :

- Path : 子结构存储路径
- Geom : 几何信息
- Nodes : 连接节点

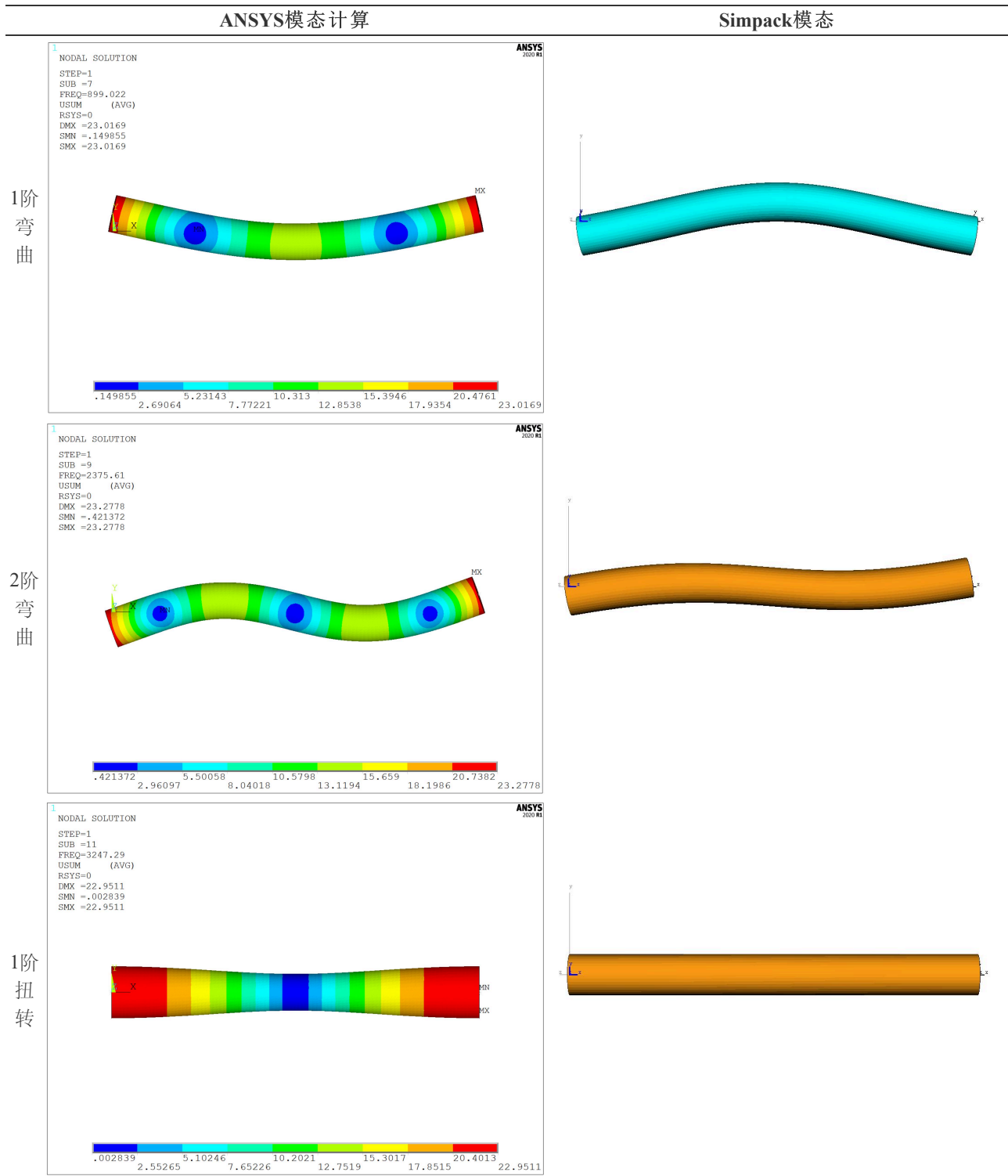
3 案例

3.1 Shaft1 compare (Rigid connection) (Flag=1)

```

1  % Shaft 1
2  inputshaft1.Length = 500;
3  inputshaft1.ID = [0,0];
4  inputshaft1.OD = [50,50];
5  paramsshift1 = struct();
6  obj1 = shaft.Commonshaft(paramsshift1, inputshaft1);
7  obj1 = obj1.solve();
8  SubAss=obj1.output.Assembly;
9  %% Define Element Types
10 ET1.name='21';ET1.opt=[3,0];ET1.R=[0,0,0,0,0,0];
11 SubAss=AddET(SubAss,ET1);
12 Acc_ET=GetNET(SubAss);
13 %% Define Connections
14 SubAss=AddCnode(SubAss,-0.001,0,0);
15 SubAss=AddMaster(SubAss,0,1);
16 SubAss=AddSlaver(SubAss,1,'face',301);
17 SubAss=SetCnode(SubAss,1,Acc_ET);
18 SubAss=SetRbe2(SubAss,1,1);
19
20 SubAss=AddCnode(SubAss,500.001,0,0);
21 SubAss=AddMaster(SubAss,0,2);
22 SubAss=AddSlaver(SubAss,1,'face',302);
23 SubAss=SetCnode(SubAss,2,Acc_ET);
24 SubAss=SetRbe2(SubAss,2,2);
25
26 NodeNum=[1;2];
27 Type={'All';'All'};
28 PartNum=[0;0];
29 Master=table(PartNum,NodeNum,Type);
30 inputSubStr.SubStr=SubAss;
31 inputSubStr.Master=Master;
32 paramsSubStr.Name="Shaft1";
33 Sub = solve.SubStr(paramsSubStr,inputSubStr);
34 Sub = Sub.solve();
35 Plot3D(Sub)
36 FbiGenerate(Sub)
37
38 Multi=MultiBody('Shaft1 Test');
```

当采用刚性连接时，使用SetRbe2建立刚性区域，设置完成后，使用Buffalo SubStr模块导出超单元，并生成fbi文件。对比ANSYS和Simpac导入模态的结果，可以发现二者基本一致。



以下分别为ANSYS和Simpack的计算结果，两者的误差非常小。

***** INDEX OF DATA SETS ON RESULTS FILE *****

SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE
1	0.0000	1	1	1
2	0.0000	1	2	2
3	0.0000	1	3	3
4	0.0000	1	4	4
5	0.82913E-03	1	5	5
6	0.98925E-03	1	6	6
7	899.02	1	7	7
8	899.16	1	8	8
9	2375.6	1	9	9
10	2375.9	1	10	10
11	3247.3	1	11	11
12	4411.2	1	12	12
13	4411.7	1	13	13
14	5126.7	1	14	14
15	6495.1	1	15	15
16	6851.7	1	16	16
17	6852.9	1	17	17
18	9579.7	1	18	18
19	9581.6	1	19	19
20	9745.8	1	20	20
21	10238.	1	21	21
22	12507.	1	22	22
23	12508.	1	23	23
24	12999.	1	24	24

Body Properties: \$B_Body1

Name: \$B_Body1
Description:
Type: Linear flexible

FE Properties
Mass Properties
Position
Modes
Options
Loads
Outputs

Eigenmode: f-min, all modes P f-min 0.1 f-max 0
Interface modes: IRM P f-exc 0
Transformation Frequency: None P f-tr 0
Cut-Off Frequency: None P f-cut 0
Damping type: Auto P

	Description	Damping	Position	Velocity	Pre-Stress Forces	Scaling Factor	Dependent	Dep. (active)	Active
1	Eigen Mode 7: 899.034 Hz	0.02	0	0	0	1	automatic	independent	yes
2	Eigen Mode 8: 899.170 Hz	0.02	0	0	0	1	automatic	independent	yes
3	Eigen Mode 9: 2375.772 Hz	0.02	0	0	0	1	automatic	independent	yes
4	Eigen Mode 10: 2376.092 Hz	0.02	0	0	0	1	automatic	independent	yes
5	Eigen Mode 11: 3247.607 Hz	0.02	0	0	0	1	automatic	independent	yes
6	Eigen Mode 12: 4412.456 Hz	0.02	0	0	0	1	automatic	independent	yes
7	Eigen Mode 13: 4412.955 Hz	0.02	0	0	0	1	automatic	independent	yes
8	Eigen Mode 14: 5128.859 Hz	0.02	0	0	0	1	automatic	independent	yes
9	Eigen Mode 15: 6497.121 Hz	0.02	0	0	0	1	automatic	independent	yes
10	Eigen Mode 16: 6855.263 Hz	0.02	0	0	0	1	automatic	independent	yes
11	Eigen Mode 17: 6856.487 Hz	0.02	0	0	0	1	automatic	independent	yes
12	Eigen Mode 18: 9591.331 Hz	0.02	0	0	0	1	automatic	independent	yes
13	Eigen Mode 19: 9593.269 Hz	0.02	0	0	0	1	automatic	independent	yes
14	Eigen Mode 20: 9754.733 Hz	0.02	0	0	0	1	automatic	independent	yes
15	Eigen Mode 21: 10250.448 Hz	0.02	0	0	0	1	automatic	independent	yes
16	Eigen Mode 22: 12527.981 Hz	0.02	0	0	0	1	automatic	independent	yes
17	Eigen Mode 23: 12529.582 Hz	0.02	0	0	0	1	automatic	independent	yes
18	Eigen Mode 24: 13016.051 Hz	0.02	0	0	0	1	automatic	independent	yes

Frames/Period: 30 Frame: 17 Scaling: 1.000000 Mode:
Filename: D:\002_RoTA_Testing\Test_121_SubStr\Shaft1 Test-mod.output\Shaft1_Test_mod.mp4
Settings...

Comment

OK Cancel Apply

3.2 Shaft1 compare (Flexible connection) (Flag=2)

```
1  % Shaft 1
2  inputshaft1.Length = 500;
3  inputshaft1.ID = [0,0];
4  inputshaft1.OD = [50,50];
5  paramsshft1 = struct();
6  obj1 = shaft.Commonshaft(paramsshft1, inputshaft1);
7  obj1 = obj1.solve();
8  SubAss=obj1.output.Assembly;
9
10 mat1.table=["MU",0.15];
11 SubAss=AddMaterial(SubAss,mat1);
12 Acc_Mat=GetNMaterial(SubAss);
13
14 %% Define Element Types
15 ET1.name='173';ET1.opt=[2,2;4,1;12,5];ET1.R=[];
16 SubAss=AddET(SubAss,ET1);
17 ET2.name='170';ET2.opt=[2,1;4,11111];ET2.R=[];
18 SubAss=AddET(SubAss,ET2);
19 ET3.name='21';ET3.opt=[3,0];ET3.R=[0,0,0,0,0,0];
20 SubAss=AddET(SubAss,ET3);
21 Acc_ET=GetNET(SubAss);
22 %% Define Contacts
23 SubAss=AddCnode(SubAss,0,0,0);
24 SubAss=AddCnode(SubAss,500,0,0);
25 SubAss=SetCnode(SubAss,1,Acc_ET);
26 SubAss=SetCnode(SubAss,2,Acc_ET);
27
28 SubAss=AddCon(SubAss,1,301);
29 SubAss=AddTar(SubAss,1,0,1);
30 SubAss=SetConMaterial(SubAss,1,Acc_Mat);
31 SubAss=SetConET(SubAss,1,Acc_ET-2);
32 SubAss=SetTarET(SubAss,1,Acc_ET-1);
33
34 SubAss=AddCon(SubAss,1,302);
35 SubAss=AddTar(SubAss,2,0,2);
36 SubAss=SetConMaterial(SubAss,2,Acc_Mat);
37 SubAss=SetConET(SubAss,2,Acc_ET-2);
38 SubAss=SetTarET(SubAss,2,Acc_ET-1);
39
40 NodeNum=[1;2];
41 Type={'A11';'A11'};
42 PartNum=[0;0];
43 Master=table(PartNum,NodeNum,Type);
44 inputSubStr.SubStr=SubAss;
45 inputSubStr.Master=Master;
46 paramsSubStr.Name="Shaft1";
47 Sub = solve.SubStr(paramsSubStr,inputSubStr);
48 Sub = Sub.solve();
49 Plot3D(Sub)
50 FbiGenerate(Sub)
51
52 Multi=MultiBody('Shaft1 Test');
53 Multi=AddBody(Multi,Sub);
54
```

采用柔性连接时，不能直接采用SetRbe3建立连接，Simpack不识别ANSYS中此连接。需采用建立接触对的方式建立柔性连接，设置Contact属性为ET1.opt=[2,2;4,1;12,5]，接着设置Target属性ET2.opt=[2,1;4,111111]，此时的Marker点为ANSYS中的从节点。

在实际工程中需要依据经验和测试的结果来决定连接采用柔性还是刚性，比如螺栓连接，相对较刚，可采用刚性连接，再比如轴承，如果直径较大，建议采用柔性连接。

4 参考文献