KATEDRA
VÝROBNÍCH STROJŮ
A KONSTRUOVÁNÍ

FAKULTA STROJNÍ
VŠB–TU OSTRAVA

17. listopadu 15, 708 33 Ostrava - Poruba, Česká republika
Tel.: +420 597 324 467, Fax: +420 324 559
E-mail: marcela.dluhosova@vsb.cz
Web: www.340.vsb.cz

1 / 3

NX, FEM

Description for images: number of description = file name of the corresponding picture.
For default NX environment settings.
Next continuing document: „nx-modelling".

1. 2.
Start of work in Modeler. Open file from folder.

3.
3D model is in layer 100 with name of Category: ORIGINAL PART.
The 3D model will be used as default input geometry for next FEM simulation.

4.
Go to application: Advanced Simulation.
FEM: solid body definition / mesh, material (preprocessing)
SIM: boundary condition, solving, resaults, (processing, postprocessing)

5.
Introduction display in Advanced Simulation.
Left bookmarks menu: Simulation Navigator and Simulation Part View, Part Navigator

6. 7.
Creating a new FEM file.
NX Nastran: solver.
Folder and filename FEM.

8.
Creating a new FEM file.
Next options.
Create Idealized Part [x]
Bodies: None: in this phase without 3D geometry.

9.
Environment of FEM in application Advanced Simulation, after FEM file definition.

10. 11. 12.
Save all related files.
I.PRT: idealized part (geometry)
FEM: simulation - mesh

12.
On left side:
Simulation Navigator: items of FEM simulatin
Simulation File View: 2-click go to file .PRT, I.PRT, .FEM.

13.
2-click on I.PRT - information about use to function WAVE. This is a copy of geometry in general work layout.

13.
Use the function WAVE for selected 3D geometry.

KATEDRA
VÝROBNÍCH STROJŮ
A KONSTRUOVÁNÍ

FAKULTA STROJNÍ
VŠB–TU OSTRAVA

17. listopadu 15, 708 33 Ostrava - Poruba, Česká republika
Tel.: +420 597 324 467, Fax: +420 324 559
E-mail: marcela.dluhosova@vsb.cz
Web: www.340.vsb.cz

2 / 3

14.
Basic work layout: i-part
Laout 100: input original part
Hide the layout 100: displaing i-part only. It is as same view as original part.

15. 16. 17.
Simulation File View:
Original part: empty display, because layout 100 is hidden.
I-part: display of 3D part
FEM: empty display, because geometry here was not defined from i-part.

18.
Simulation File View – model1_fem1_i – 3D geometry for idealization.

19. 20. 21.
Idealize Geometry:
Select of 3D geometry (part) pro idealization.
Setting parameters for hide:
- small holes
- edge blends

22. 23. 24. 25.
Split Body:
Select of 3D object
Features and parameters for plane definition.

26. - 35.
Divide of volume geometry by extrude circle.
Go to Modeling.
Part Navigator: history of idealizations.
Sketch on the plane and finish sketch.
Split Body / Extrude and select volume for dividing.
Go to Advanced Simulation.
Idealized part is done.

36.
FEM / Simulation File View: go to model1_fem1
Geometry: None
Simulation Navigator: mouse right button - menu - Edit.

37. – 40.
Select volume parts and OK.
In the FEM is 3D geometry for next work, mesh creating.

41. – 43.
Parameters and creating of mesh.
Function 3D Swept Mesh:
linear 6-sides elements on the selected volume, element size parameter.
OK and mesh will be created.

44. -45.
Parameters and creating of mesh.
3D Tetrahedral: 4-sides elements and size parameter: 5.
OK and mesh will be created.

46. - 47.
Before create mesh for last volume part.
Mash Mating: select all 3 volume parts and confirm. It is setting for elements continue between neighboring meshes.
Mash Mating is in history tree.

48. – 52.
Create mesh for last volume part.
Linear 4-sides element, size parameter: 10.
Transition with Pyramid Elements: confirm for 6-sides elements continuing to pyramid elements.
Pyramid elements can continue to 4-sides elements.
Mesh update.

53. – 58.
Material – Physical Properties.
Geometry is defined by mesh.
Physical Properties: define of the material for object PSOLID1 – it is all mesh.
Material is possible use from share NX database: NX - Library Materials.
Or it is possible to create a new one – Local Materials.

59.
FEM: define volume part through:
Mesh (geometry)
Material parameters (material)
All is in FEM history - Simulation Navigator.

Next step is go to SIM (simulation).