NX Nastran on HPC

What is NX NASTRAN?

Siemens NX Nastran is a finite element analysis (FEA) software package used for structural analysis and simulation. It is part of the Siemens NX software suite, which is a computer-aided design (CAD) and computer-aided engineering (CAE) software platform used for product design, engineering, and manufacturing.

NX Nastran is a widely used FEA software in the aerospace, automotive, and defense industries, and is used to simulate the behavior of structures, systems, and components under various conditions, such as static loads, dynamic loads, thermal loads, and vibrations. It can be used to analyze complex geometries and materials and can simulate linear and nonlinear behavior.

NX Nastran offers a wide range of analysis capabilities, including linear static, modal, dynamic response, non-linear static, non-linear transient, and buckling analysis. It also includes advanced features for optimization and design exploration, such as topology optimization and shape optimization, which can help engineers design more efficient and lightweight products.

Links:

Official Website

User Guides

Versions Available:

The following versions are available on the cluster:

- NX Nastran 2019.1
- NX Nastran 12.0

How to load NX NASTRAN?

To load NX NASTRAN, use the following commands:

```
#Load the NX Nastran module
module load nxnastran/12
```

To verify if the module is loaded correctly, use the following command,

```
# List all the module loaded in the environment
module list
```

In a fresh environment, this only shows NX Nastran module loaded.

How to use NX NASTRAN?

Here is a link to the operational manual for NX Nastran:

[Operational Manual]

To use Siemens NX Nastran, you will need to follow these general steps:

1. Prepare the model: The first step is to create or import the 3D geometry of the model you want to analyze. This may involve using CAD software, such as

Siemens NX, to create the model or importing a model from another software format.

2. Define the mesh: The next step is to define the mesh of the model, which involves dividing the model into a series of small elements that can be analyzed using finite element analysis. Nastran provides various meshing tools and methods to help you create an accurate and efficient mesh.
3. Assign material properties: Once the mesh is defined, you need to assign material properties to the model. This may involve specifying the material type, density, modulus of elasticity, and other relevant properties.
4. Apply boundary conditions: After the materials are assigned, you can apply the boundary conditions to the model. This involves defining the forces, pressures, temperatures, or other loading conditions that act on the model.
5. Configure the analysis: The next step is to configure the analysis settings, such as the type of analysis, solver options, and convergence criteria.
6. Run the analysis: Once the analysis is set up, you can run the simulation and monitor its progress. Depending on the simulation's complexity, this may take a lot of time and computational resources.
7. Analyze the results: After the analysis is complete, you can analyze the results to understand the behavior of the model under the given loading conditions. This may involve visualizing the results, generating reports, or comparing the results to design specifications.

To submit a job, use the following script as a reference,

```
#!/bin/bash
#SBATCH -J my_job         # Job name
#SBATCH -o my_job.out     # Output file name (%j will be replaced by
job ID)
#SBATCH -e my_job.err     # Error file name (%j will be replaced by
job ID)
#SBATCH -N 1              # Number of nodes
#SBATCH -n 8              # Number of cores per node
#SBATCH -t 01:00:00       # Max time limit (hh:mm:ss)
#SBATCH --mem=16G         # Memory per node (total memory = N nodes *
memory per node)
#SBATCH -p main       # Partition name

# Load the Nastran module
module load nxnastran/12

# Change to the working directory
```

```
cd /path/to/working/directory

# Run the simulation
nxnastran my_input_file.dat

# Copy the results to the output directory
cp my_results_file.f06 /path/to/output/directory
```

Where to find help?

If you are confused or need help at any point, please contact OIT at the following address.

https://ua-app01.ua.edu/researchComputingPortal/public/oitHelp