Microsystems 1.1

Kunal D. Patil
(kdp@mecheng.iisc.ernet.in)
CoNe Lab, IISc

November 17, 2014

Contents

—1—

Introduction to `Microsystems` module

This document serves as a brief introduction to `Microsystems` (version `1.1`) module in `NISA`. Figure (1.1) shows the snapshot of `Microsystems` module implemented in the `DISPLAYIV` GUI of `NISA`.
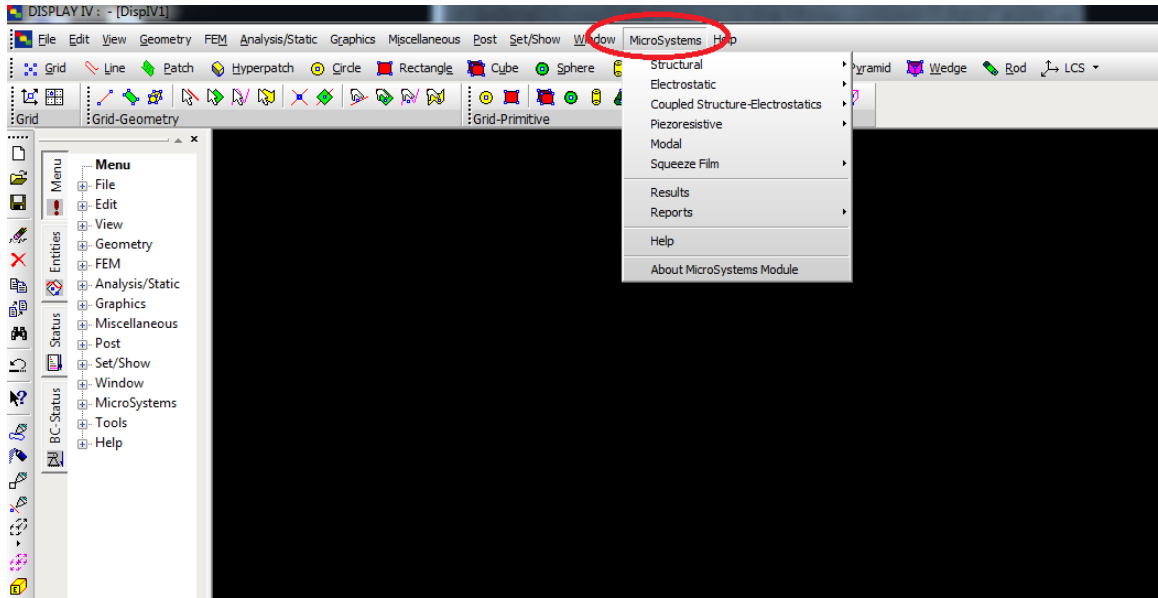

Figure 1.1. `Microsystems` module in `NISA`

After successful installation of `Microsystems` module in the `DISPLAYIV` of `NISA`, the version `1.1` is ready for working with the following analyses;

- Structural analysis (static and transient)
- Electrostatic analysis
- Coupled structure-electrostatic analysis (static and transient)
- Squeeze film analysis (harmonic)

Each of the above mentioned analysis type is discussed briefly in the following sequence [1];

- **Pre-processing** (creating geometry and suitable mesh, specifying material data, boundary conditions, pressure, point loads etc.).
- **Analysis** (computation of field quantities such as stresses, strains, displacements, potential etc.).

---

[1]Squeeze film analysis is discussed in the separate document.

- **Post-processing** (visualization of field quantities and their variation with respect to other quantities (2D plots)).

—2—

Structural analysis

Figure (2.1) shows the snapshot of structural analysis section of the `Microsystems` module. There are two types of analysis;

- Static analysis
- Transient analysis

At present, structural analysis offers solution to problems with following features;

- fully three-dimensional thin and chunky geometries
- geometrically non-linear (large displacement, small strain) behaviour
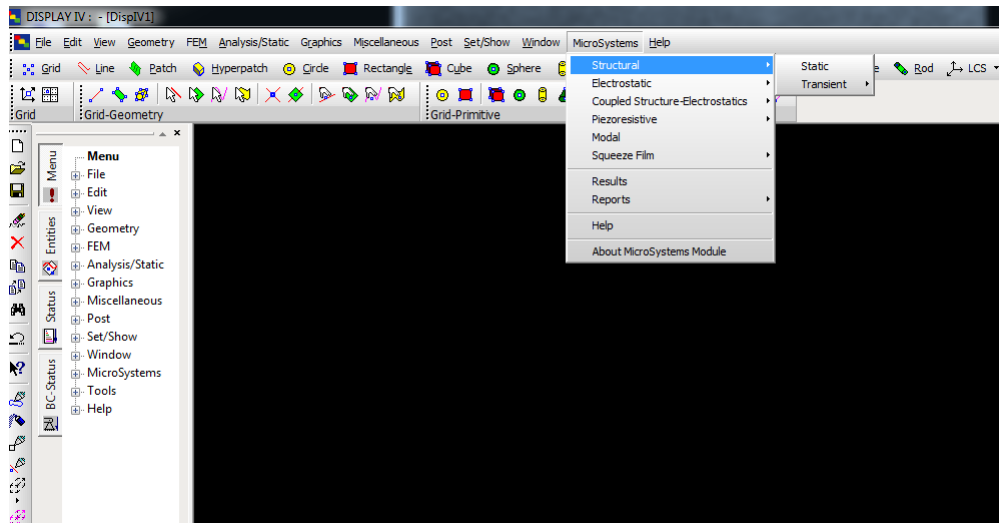- linear elastic

Figure 2.1. Structural analysis section in the `Microsystems` module

2.1   Pre-processing

This section describes how to create problem geometry, FE meshing, prescribing boundary (displacements, point and pressure loads) conditions and material data specifications for the structural analysis. Description given here is very

specific to the structural analysis section implemented in the `Microsystems` module.[2]

Following points describe systematic procedure for pre-processing in `NISA` specific to `Microsystems` structural analysis;

- **Geometric modelling**:

    There is no restriction on using various ways of creating geometry in `NISA` for structural analysis. One can follow standard procedure of Grid $\Rightarrow$ Line $\Rightarrow$ Patch $\Rightarrow$ Hyper patch or in the reverse way or using workplane and then extrusion into 3D.

    NOTE: Use **micrometer** (μm) (as default) dimension for creating geometry i.e., all dimensions are in microns. For example if the length is 80μm then create line of length 80.
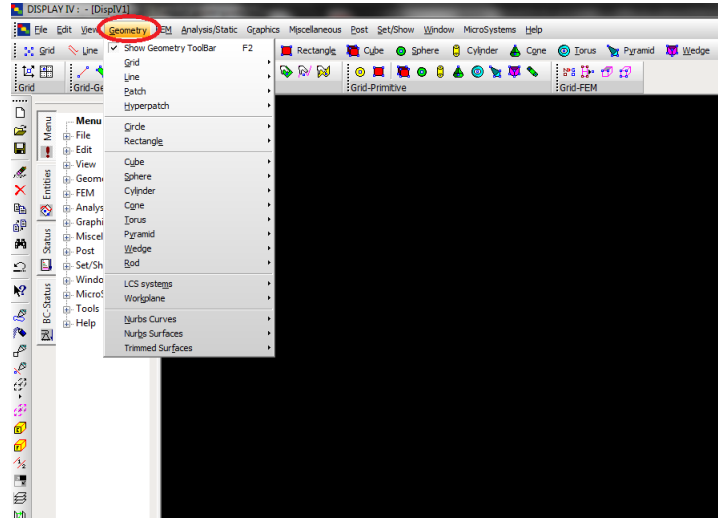

Figure 2.2. Geometry tool bar in `NISA`

- **Material data**: Once the geometry of problem is finished, material property data list has to be created. Specify all material properties in **SI units**. Figures (2.3 and 2.4) show the material property data specification in `NISA`.

    – Click `Add` button and specify `MATERIAL ID` [3]
    – `EX` : Modulus of elasticity
    – `NUXY` : Poisson's ratio
    – `DENS`: Density of material

---

[2]Arbitrary modelling and automatic mesh generation in `NISA` is not valid for `Microsystems` module. One should stick with the pre-processing procedure described in the section.

[3]Number of material id's are equal to the number of different materials involved in the problem and mentioned material properties need to be specified for all material id's respectively.
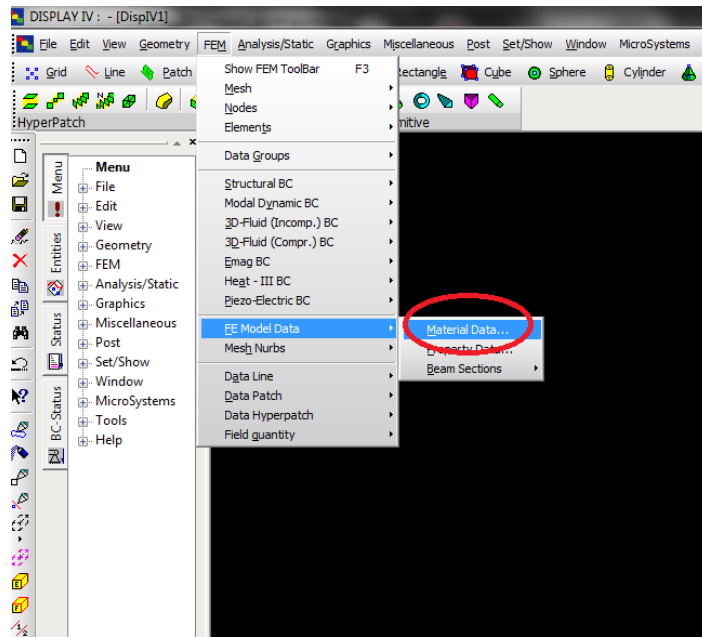
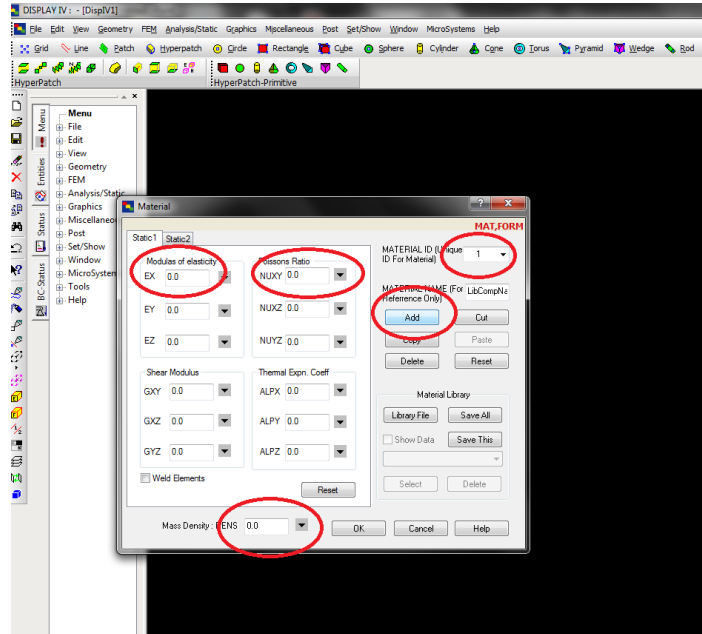**Figure 2.3.** Material property data specification in NISA

Figure 2.4. Material property data specification in NISA

- **Mesh details**:

  After material data specification, meshing is to be done for each Hyper patch. Figure (2.5) shows the FEM mesh toolbar in `NISA` used for structural analysis. For mesh specification, only `FEG` option → `Wedge` and `Hexahedron` types are used.
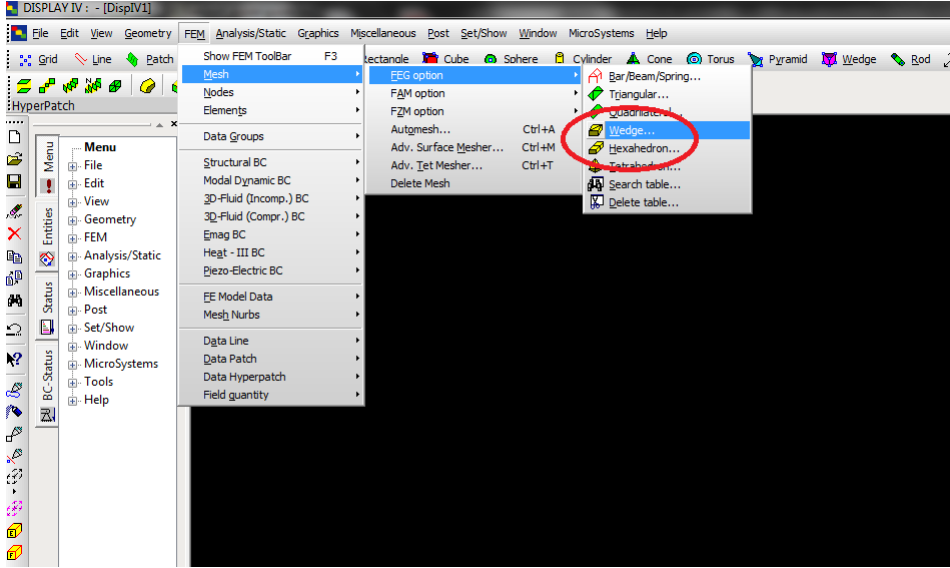

Figure 2.5. Meshing for structural analysis

- In the `Hexahedron` section, Figure (2.6), `3D Solid` is used for conventional displacement based FEM and `3D Hybrid Solid` is used for hybrid FEM. [4] [5] [6]

- In the case of `Wedge` section, Figure (2.7), `3D Solid` is used for the conventional displacement based FEM and `3D Concrete` is used for hybrid FEM. [7]

---

[4] Do not use any other element type. Only above mentioned types are used for structural analysis. Also note that, first one has to select appropriate Hyper patch by clicking `Input Hyp IDs` and then mesh it by specifying number of elements in the `E1/E2/E3` directions.

[5] While meshing each Hyper patch, do not forget specifying associated `Material ID`.

[6] It should be noted that meshing is done on Hyper patch and requirement of nodal continuity must be satisfied on Patch shared by two adjacent Hyper patches i.e., the meshing on adjacent Hyper patches if results into 9 nodes to the left and 6 node to the right of the common shared Patch then appropriate merging tolerance has to be specified to get continuity of nodes.

[7] Use only one layer of wedge elements in the radial direction. Number of elements in the 'θ' and 'z' direction can vary.
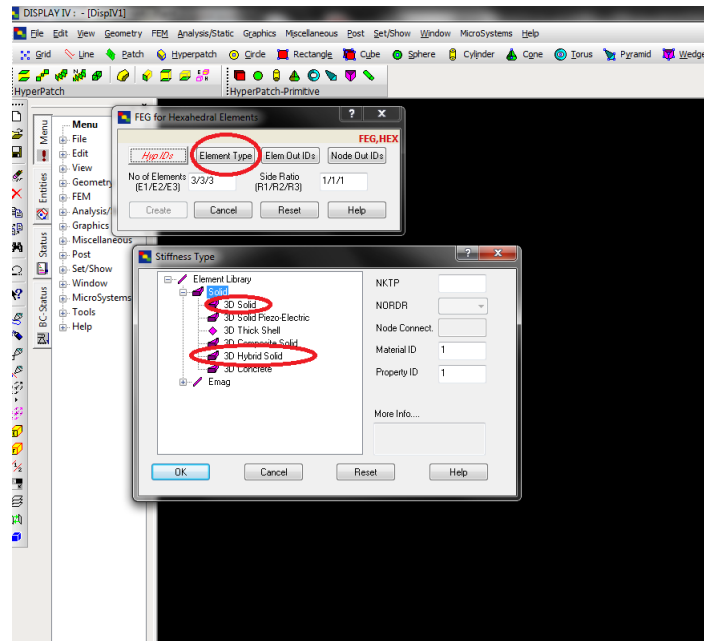
Figure 2.6. Meshing for structural analysis: Hexahedron
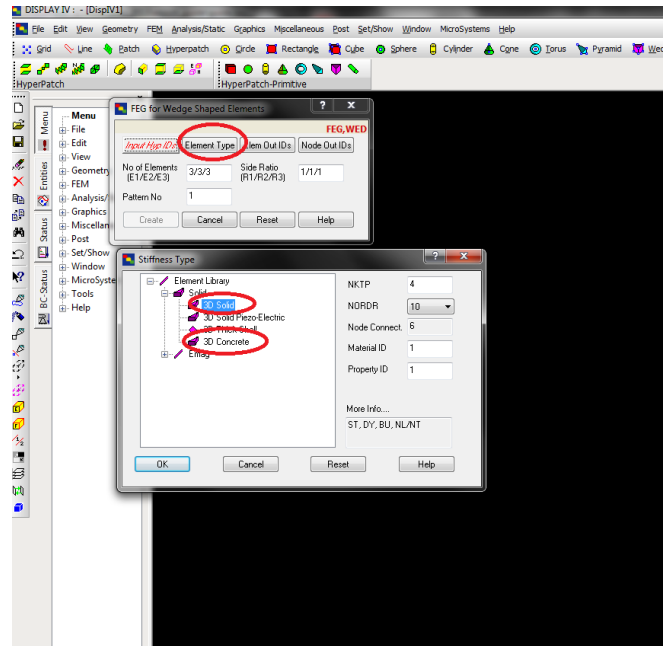

Figure 2.7. Meshing for structural analysis: Wedge

- **Structural boundary conditions**:
  There are three types of boundary conditions specified for structural analysis; [8]
    - Displacement boundary conditions (Figures (2.8) and (2.9))
    - Nodal force boundary conditions (Figures (2.10) and (2.11))
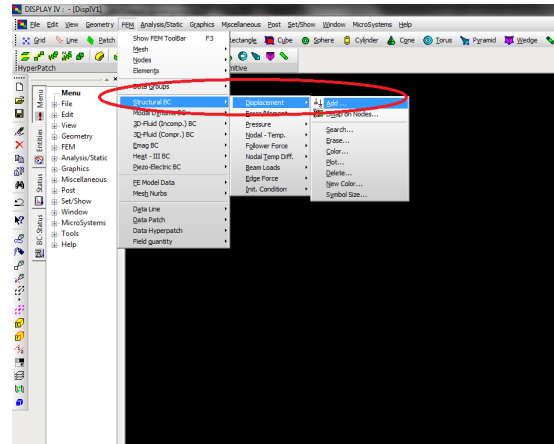   - Pressure boundary conditions (Figures (2.12) and (2.13)) [9]
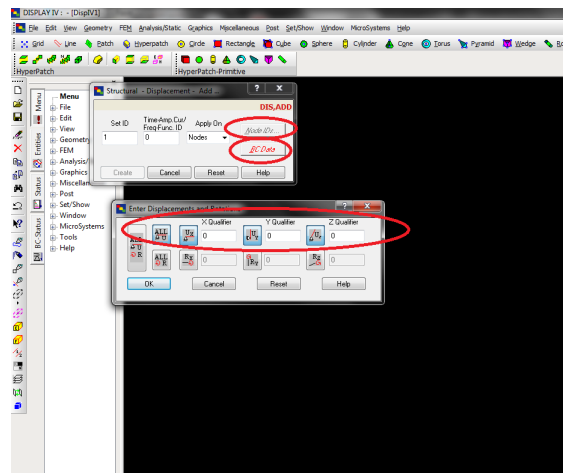

Figure 2.8. Boundary conditions: Displacement

Figure 2.9. Boundary conditions: Displacement

---

[8]Specify boundary conditions in SI units.

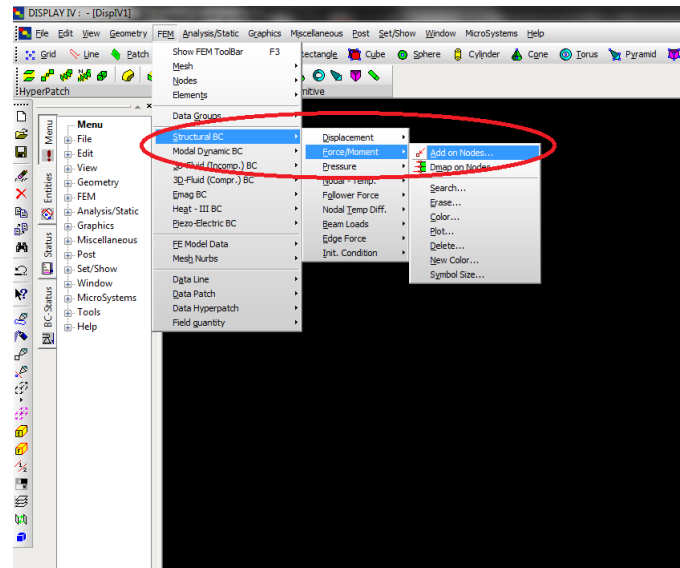[9]Choose appropriate face on which pressure is acting or specify face number manually.

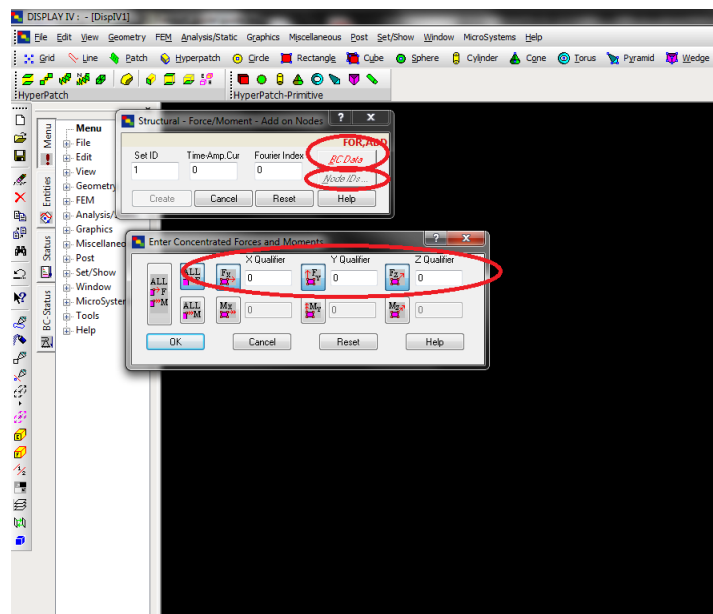**Figure 2.10.** Boundary conditions: Nodal force
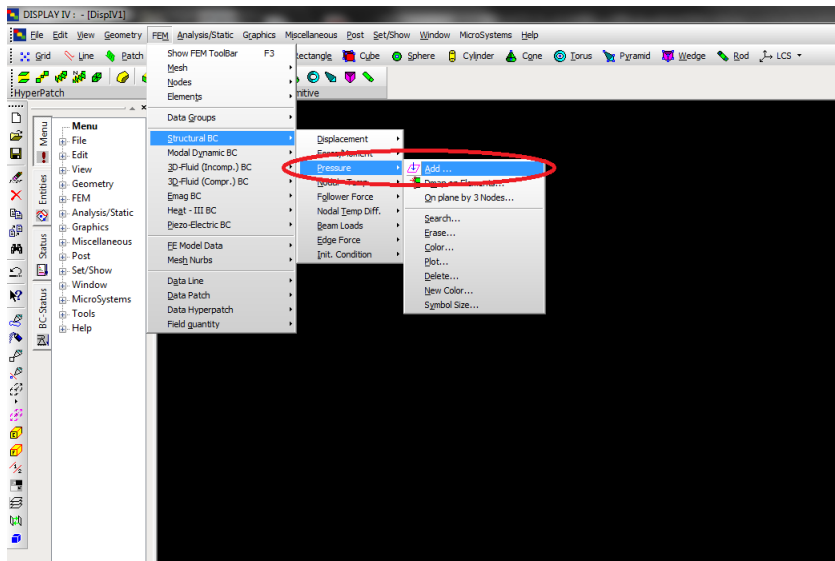

Figure 2.11. Boundary conditions: Nodal force

Figure 2.12. Boundary conditions: Pressure
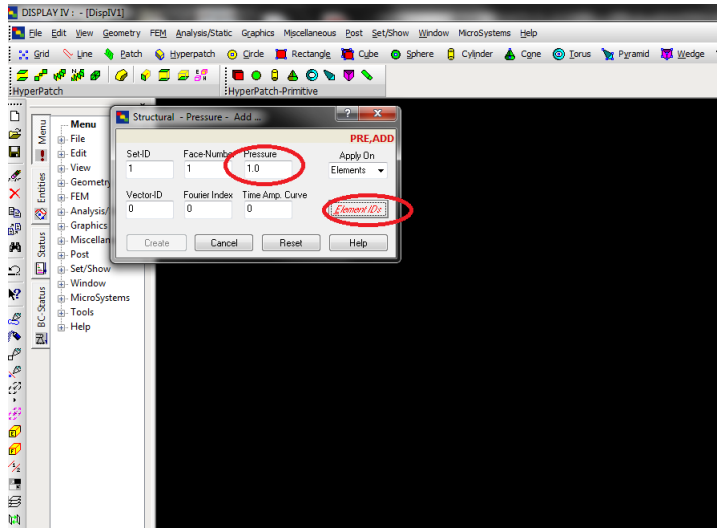

Figure 2.13. Boundary conditions: Pressure

2.2 Analysis

2.2.1 `Static` analysis

Before executing static analysis, save the file as <file>.DBS and <file>.NIS. [10]
In the case of static structural analysis, click on the section `Static` as shown
in the Figure (2.14) (if file is not open then load the <file>.NIS from `File`
menu or load using `Run` section Figure (2.15)). Specify the 'node of interest'
Figure (2.16), at which some useful information (displacement, normal stress
etc.) is required. [11] Click `OK` to start the analysis. Wait for the analysis to get
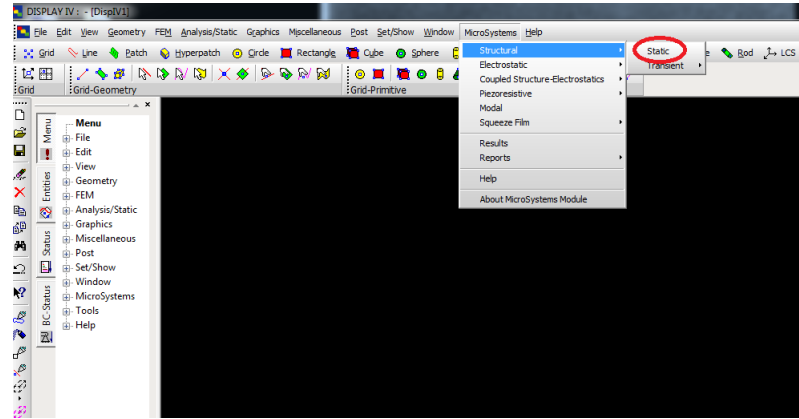completed. [12].

Figure 2.14. Structural analysis: Static

2.2.2 `Transient` analysis

Before executing transient analysis of structure, Figure (2.18), first specify the
`Time Data` as shown in the Figure (2.19).
`Starting Time` is `0.0 sec`,
`End Time` is the end of simulation time (in `sec`) and
`Time Step` represents the increment in the time (in `sec`).

Save the file as <file>.DBS and <file>.NIS else if not open, load the <file>.NIS
from `File` menu or using `Run` section by clicking `Execute` section (similar to Fig-
ure (2.15)). Click on the section `Execute` as shown in the Figure (2.20). Select
the 'node of interest' at which information is sought. Click `OK`. Wait for the
analysis to get completed.

---

[10] The default directory is `C:\NISARUN`

[11] This   information   is   used   to   create   some   2D   plots   in   the   directory
`C:\NISARUN\Microsystems\results\2dplots`.   Details   about   plots   are   discussed   in   the
Post-processing section.

[12] Generally screen gets busy once analysis starts. The end of analysis is indicated by display
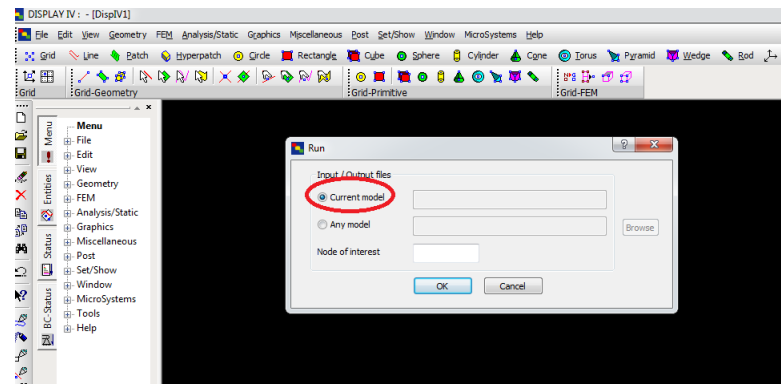of computation time, see Figure (2.17).

Figure 2.15. Loading model file from the `Run` section. By clicking on `Static`, this section is displayed.
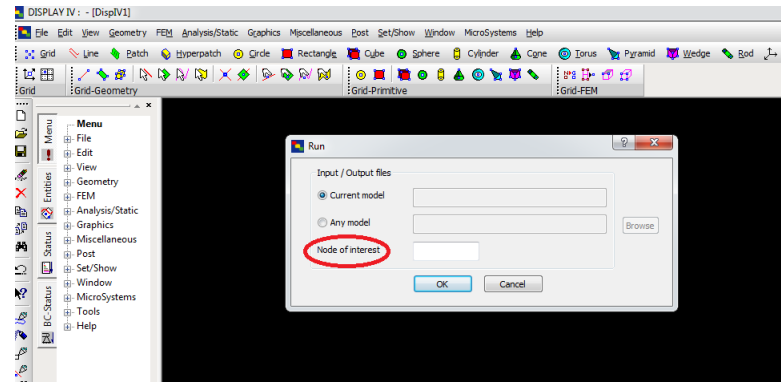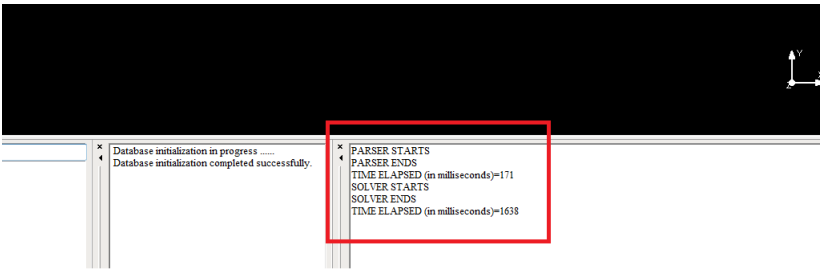

Figure 2.16. Node of interest specification
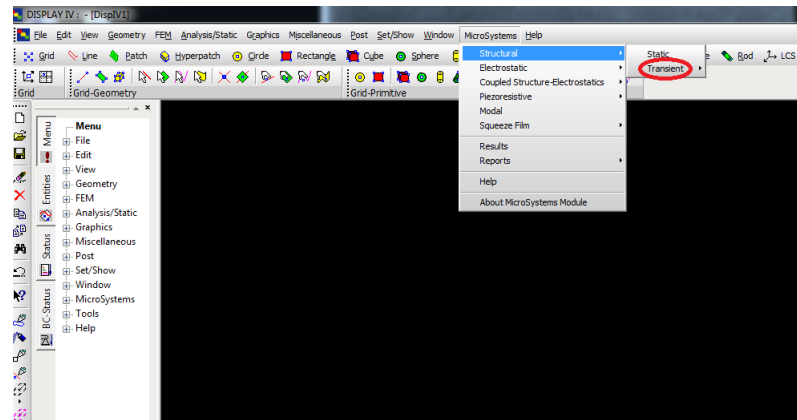

Figure 2.17. Computation time indicator

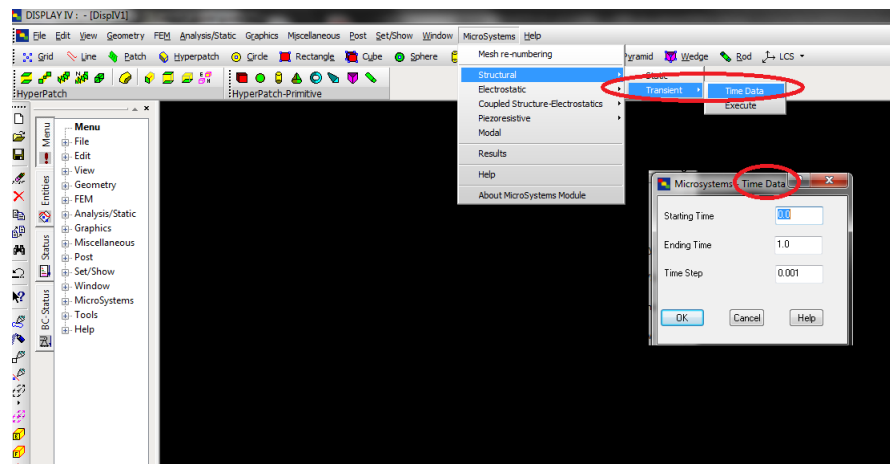**Figure 2.18.** Structural analysis: Transient
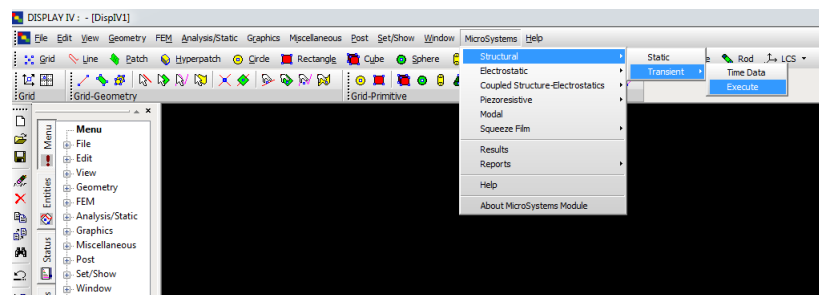

Figure 2.19. Time data

Figure 2.20. Execute

2.3 Post-processing

After completing the structural analysis, the output files are written in the directory `C:\NISARUN\Microsystems\results`. The `results` directory has following structure;

- `fields` directory
- `2dplots` directory and
- Squeeze film output files `sfdoutput.dat`, `sfdpostpro.dat` and `sfdpres.dat`.

For visualization, `fields` directory consists of information related to three basic structural field quantities,

- Displacement (`displacement.dat`)
- Normal stresses (`normal_stress.dat`)
- Shear stresses (`shear_stress.dat`)

In order to visualize displacement, load the file `displacement.dat`, from the `Results` section of `Microsystems` module. (see Figure 2.21)

For visualization of normal stresses, load the file `normal_stress.dat` from `Results` section and for shear stresses, load the file `shear_stress.dat`.

Once appropriate files are loaded from the results section, visualization of field variable is realized from the menu section as shown in the Figure (2.22).

At present, distribution of field variable over geometry and maximum and minimum values of field variables with its nodal location can be obtained from GUI.

Animation of field quantities can also be visualized using `Animation` section as shown in the Figure (2.23).
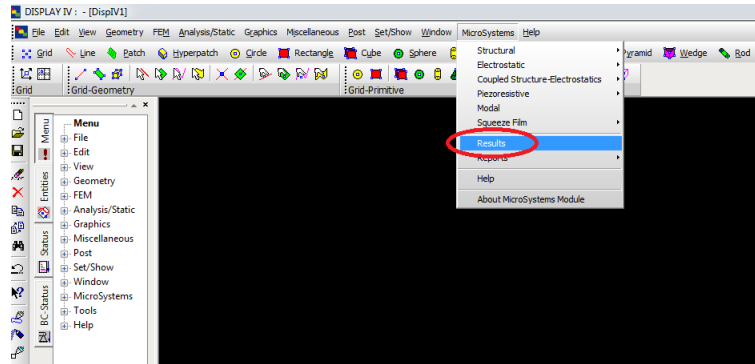
Figure 2.21. Results section of `Microsystems` module
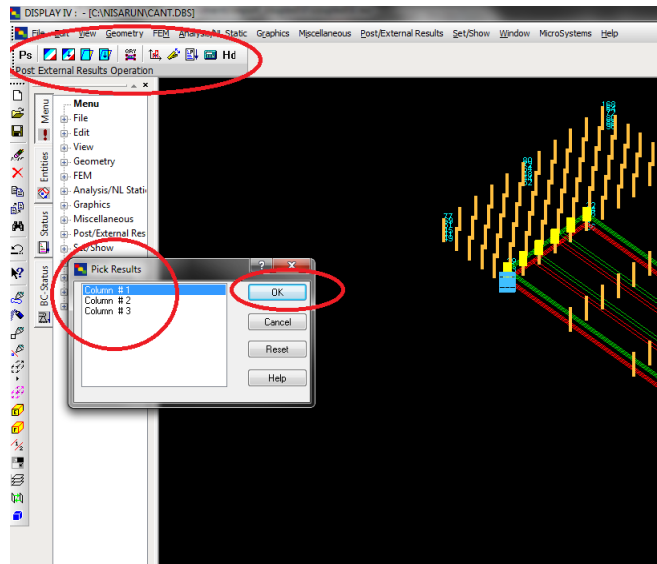
Figure 2.22. Visualization section

Figure 2.23. Animation section

The directory 2dplots, constitutes following structural analysis files;

- plotlsxx.dat (factu (factor of load applied) v/s normal stress (x dir))
- plotlsxy.dat (factu v/s shear stress (x-y dir))
- plotlsxz.dat (factu v/s shear stress (x-z dir))
- plotlsyy.dat (factu v/s normal stress (y dir))
- plotlsyz.dat (factu v/s shear stress (y-z dir))
- plotlszz.dat (factu v/s normal stress (z dir))

- `plotlux.dat` (factu v/s displacement (x dir))
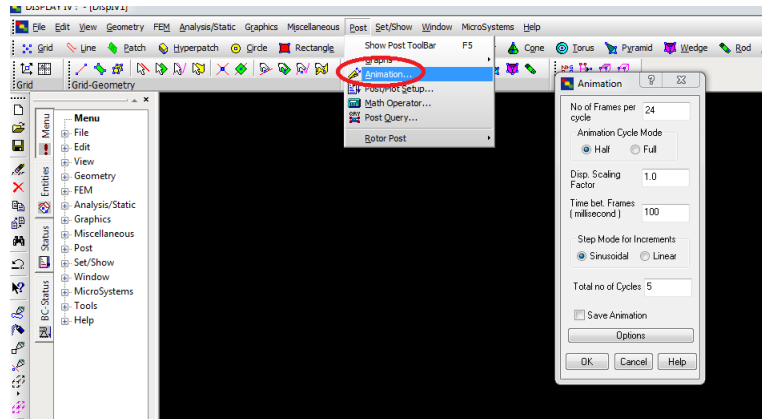- `plotluy.dat` (factu v/s displacement (y dir))
- `plotluz.dat` (factu v/s displacement (z dir))
- `plotltx.dat` (time v/s displacement (x dir))
- `plotlty.dat` (time v/s displacement (y dir))
- `plotltz.dat` (time v/s displacement (z dir))

These files represents variation of one field quantity with respect to other and can be plotted with `Graph` section as shown in the Figure (2.24).
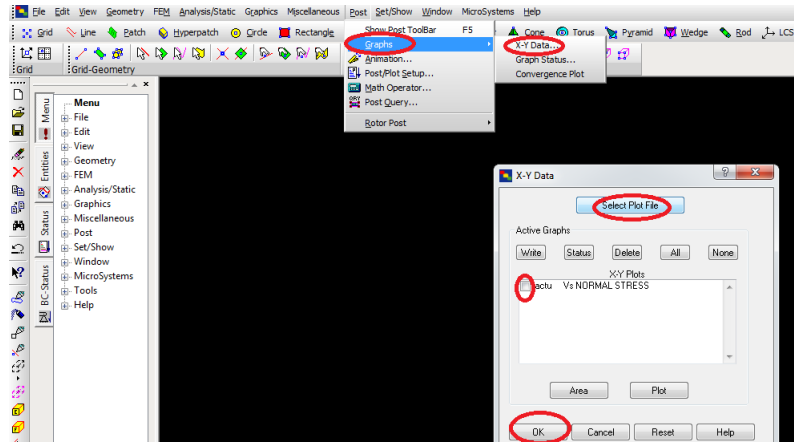

Figure 2.24. 2d plot section

2.4   Example files

Please see the video's on youtube channel `https://www.youtube.com/user/conelabiisc` (or in the directory structural analysis in `Microsystems` CD/DVD).

—3—

Electrostatic analysis

Figure (3.1) shows the snapshot of electrostatic analysis section of the `Microsystems` module.
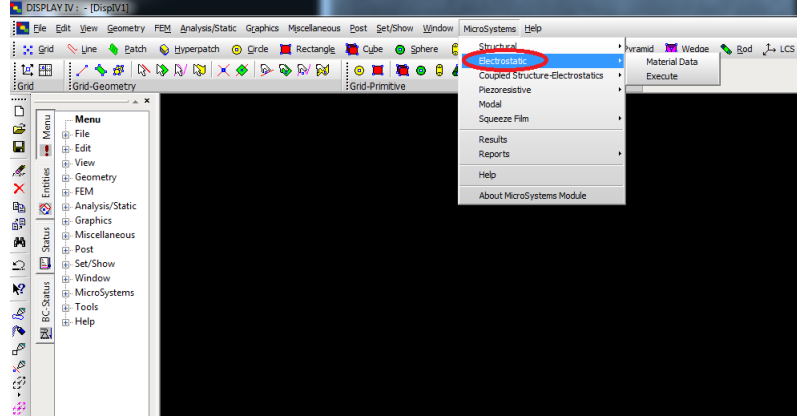

Figure 3.1. Electrostatic analysis section in the `Microsystems` module

3.1   Pre-processing

This section describes how to create geometry, mesh and giving boundary conditions for the electrostatic analysis.

Following points describe systematic procedure for pre-processing in `NISA` specific to electrostatic analysis;

- **Geometric modelling**: There is no restriction on creating geometry in `NISA` for electrostatic analysis under `Microsystems`. One can follow standard procedure of Grid $\Rightarrow$ Line $\Rightarrow$ Patch $\Rightarrow$ Hyper patch or in the reverse way or using workplane and then extrusion in to 3D.

  In the case of this analysis, as electrostatic field is involved, bounding box (of suitable dimensions) needs to be created enclosing entire model geometry. This is very important for accuracy of FE solution. Variation of electrostatic field over the enclosing domain is then computed during analysis. [13]

  NOTE: For electrostatic analysis in `Microsystems` module, use **micrometer** (μm) (as default) dimension for creating geometry i.e., all dimensions

---

[13]Creating bounding box surrounding the problem geometry is very involved process. Considerable amount of time is spent on this process and on meshing these hyperpatches. Surrounding domain is always partially conducting and partially dielectric elastic medium for example, air. See the required material properties specification. Also, see example video files for how to create and mesh such geometries.

are in microns. For example if the length is 80μm then create line of length 80.

- **Material Data**: Once the geometry of problem is finished, material ID list has to be created. This material ID specification is important to distinguish between different materials involved in the analysis. Figures ((3.2) and (3.3)) show the material ID specification for electrostatic analysis. Mechanical material properties can be set to zero for electrostatic analysis.
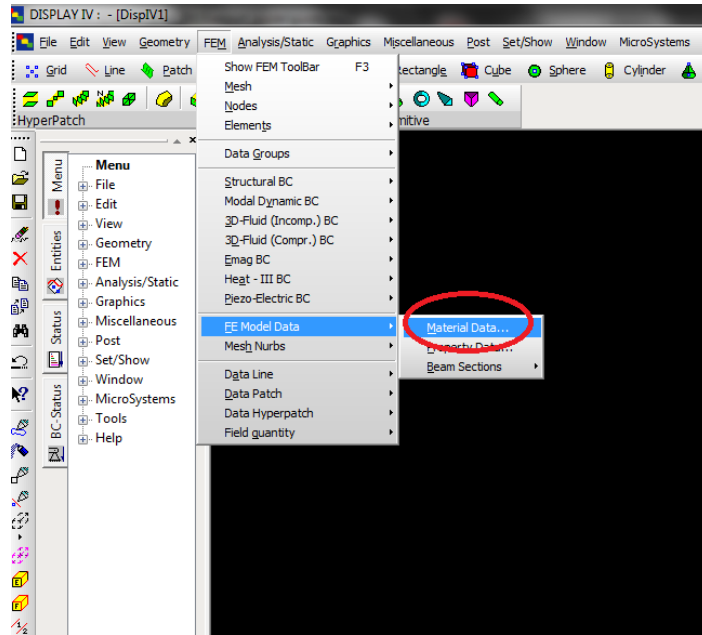

Figure 3.2. Material ID specification in `NISA`

In the case of electrostatic analysis main set of material property data involving `Relative Permittivity`, `Electrical Conductivity` needs to be specified for each material. For this specification ref Figure (3.4). It must be noted that upto only **five** different types of materials can be used in the problem. Thus, it will results into maximum of **5 Material ID's**. Use all properties in `SI` units.

- **Mesh Details**: After material data specification, meshing is to be done for each Hyper patch. Figure (3.5) shows the FEM mesh toolbar in `NISA` used for electrostatic analysis. For mesh specification, only `FEG` option $\rightarrow$ `Wedge` or `Hexahedron` types are used. [14]

---

[14]General modelling in `NISA` and automatic mesh generation is not valid for `Microsystems` module. One should stick with the pre-processing procedure described in the section.
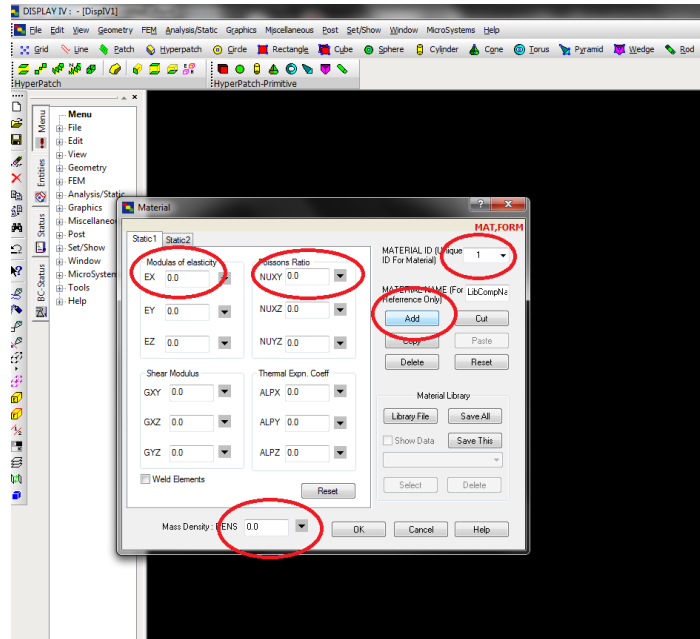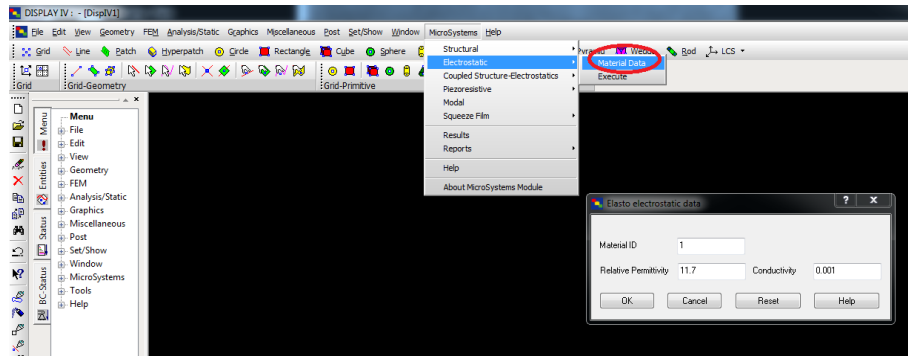
Figure 3.3. Click `Add` to create material IDs

Figure 3.4. Material property data specification in `NISA`

– In the `Hexahedron` section, Figure (3.6), `3D Solid` is used for conventional displacement based FEM. [15] [16] [17]

– In the `Wedge` section, Figure (4.8), `3D Solid` is used for the conven-

---

[15] Do not use any other element type. Only above mentioned types are used for electrostatic analysis. Also note that first, one has to select appropriate Hyper patch by clicking `Input Hyp IDs` and then mesh it by specifying number of elements in the `E1/E2/E3` directions

[16] While meshing each Hyper patch, do not forget specifying associated `Material ID`.

[17] In electrostatic analysis use of hybrid elements is of no use over conventional elements.
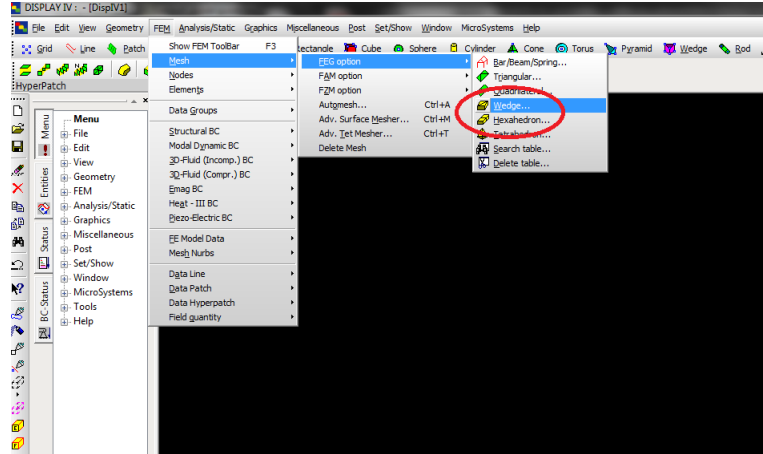
tional displacement based FEM. [18]

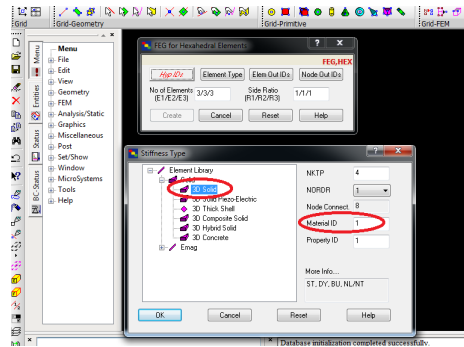**Figure 3.5.** Meshing for electrostatic analysis

Figure 3.6. Meshing for electrostatic analysis: Hexahedron

- **Boundary Conditions**: There is only one type of boundary condition need to be specified for electrostatic analysis; [19]

  – Applied potential/voltage (Figure (3.8))

---

[18]It should be noted that meshing is done on Hyper patch and requirement of nodal continuity must be satisfied on Patch shared by two adjacent Hyper patches i.e., the meshing on adjacent Hyper patches if results into 9 nodes to the left and 6 node to the right of the common shared Patch then appropriate merging tolerance has to be specified to get continuity of nodes.

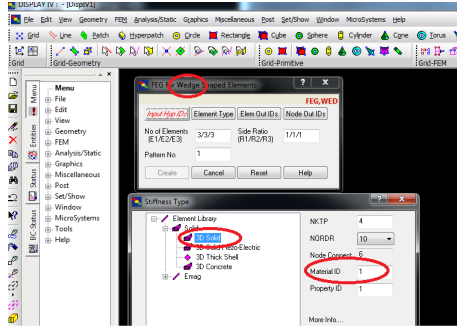[19]Specify boundary conditions in SI units.

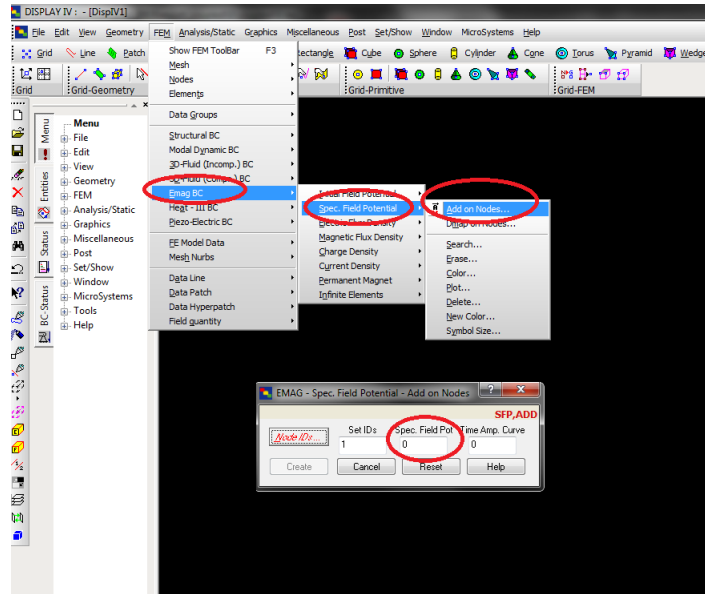**Figure 3.7.** Meshing for electrostatic analysis: Wedge

Figure 3.8. Applied potential specifications

3.2 Analysis

Before starting analysis save the file as <file>.DBS and <file>.NIS. In the case of electrostatic analysis (if not open, load the <file>.NIS from `File` menu or load using `Run` section by clicking `Execute`). Select 'node of interest' and click `OK` for analysis.

Wait for the analysis to get completed.

3.3 Post-processing

After completing analysis, output field `potential` can be graphically visualized over the problem geometry. The file `potential.dat` will be created in the
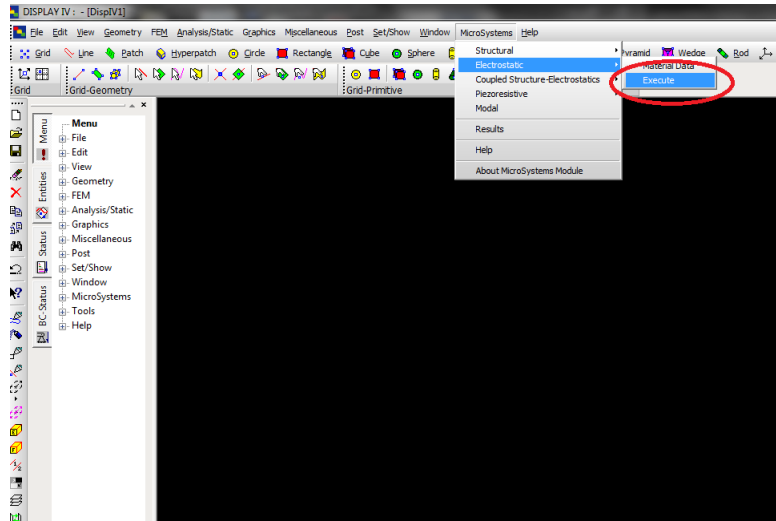
Figure 3.9. Electrostatic analysis

directory `C:\NISARUN\Microsystems\results`.

Once `potential.dat` file is loaded from the `Results` section, visualization can be realized from the menu section as shown in the Figure (2.22).

At present, maximum and minimum values of potential, and the corresponding nodal location can be obtained in GUI.

3.4 Example files

Please see the video's on youtube channel `https://www.youtube.com/user/conelabiisc` (or in the directory electrostatic analysis in `Microsystems` CD/DVD).

—**4**—

Coupled Structure–Electrostatics **analysis**

Figure (4.1) shows the snapshot of coupled structure-electrostatic analysis section of the `Microsystems` module. There are two types of analysis;

- Static analysis
- Transient analysis

At present, coupled structure-electrostatic analysis can be used for problems with following features;

- fully three dimensional thin and chunky geometries
- geometrically non-linear (large displacement, small strain) behaviour
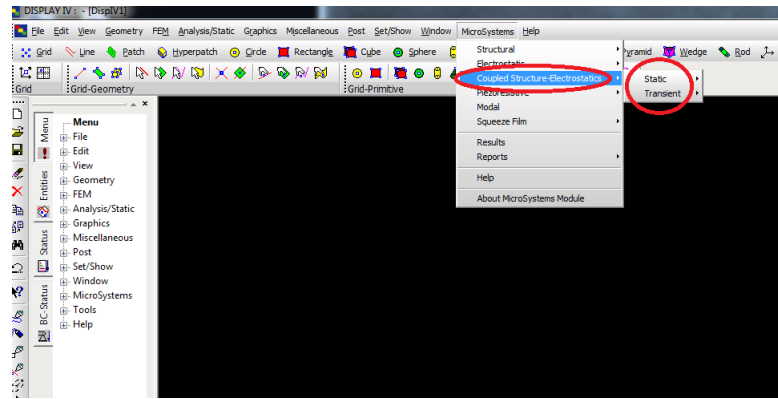- linear elastic

Figure 4.1. Coupled structure-electrostatic analysis section in the `Microsystems` module

4.1   Pre-processing

This section describes how to create geometry, mesh and giving boundary conditions (displacement, voltage etc.) for the coupled structure-electrostatic analysis. Description given here is very specific to this analysis of `Microsystems` module.

Following points describe systematic procedure for pre-processing in `NISA` specific to coupled analysis;

- **Geometric modelling**: There is no restriction on creating geometry in `NISA` for coupled analysis under `Microsystems`. One can follow standard procedure of Grid $\Rightarrow$ Line $\Rightarrow$ Patch $\Rightarrow$ Hyper patch or in the reverse way or using workplane and then extrusion in to 3D.

- In the case of this analysis, as electrostatic field is involved, bounding box (of suitable dimensions) needs to be created enclosing entire model geometry. This is very important for accuracy of FE solution. Variation of electrostatic field over the enclosing domain is then computed during analysis. [20]

  NOTE: For coupled structure-electrostatic analysis in `Microsystems` module, use **micrometer** (μm) (as default) dimension for creating geometry i.e., all dimensions are in microns. For example if the length is 80μm then create line of length 80.
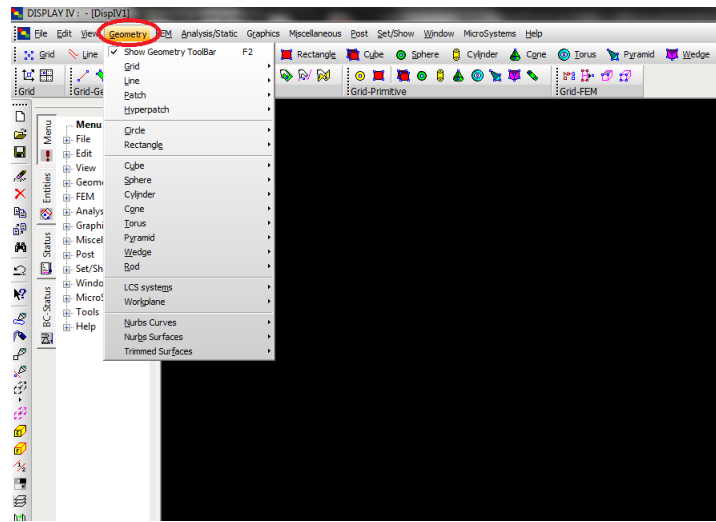

Figure 4.2. Geometry tool bar in `NISA`

- **Material Data**: Once the geometry of problem is created, material property data list has to be specified. [21] Figures ((4.3) and (4.4)) show the material property data specification in `NISA`.
  - Click on `Add` button and specify `MATERIAL ID` [22]
  - `EX` : Modulus of elasticity
  - `NUXY` : Poisson's ratio
  - `DENS`: Density of material

---

[20]Creating bounding box surrounding the problem geometry is very involved process. Considerable amount of time is spent on this process and on meshing these hyperpatches. Surrounding domain is always partially conducting and partially dielectric elastic medium for example, air. See the required material properties specification. Also, see example video files for how to create and mesh such geometries.

[21]Specify all material properties in SI units

[22]Number of material id's are equal to the number of different materials involved in the problem, and mentioned material properties need to be specified for all material id's respectively.
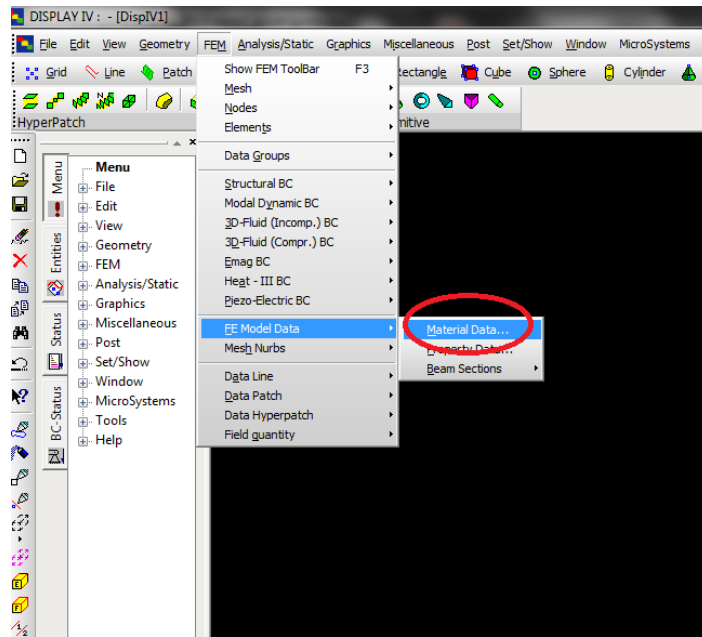
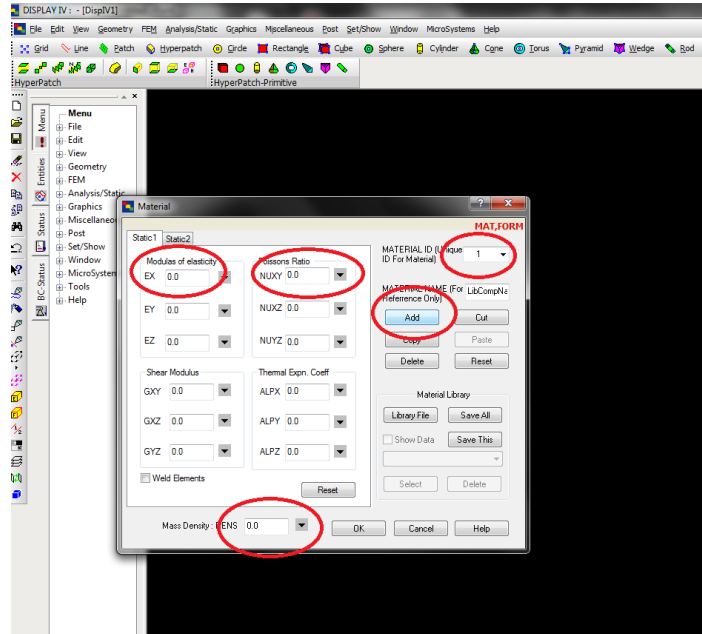**Figure 4.3.** Material property data specification in NISA

Figure 4.4. Material property data specification from NISA

- In the case of coupled structure-electrostatic analysis other set of material property data involving `Relative Permittivity`, `Electrical Conductivity` needs to be specified. For this specification refer Figure (4.5). It must be noted that upto only **five** different types of materials can be used in the problem. Thus it will results into maximum of **5 Material ID's**. Use all properties in `SI` units. For `Transient Analysis` use the corresponding `Material Data` specifications section. [23]
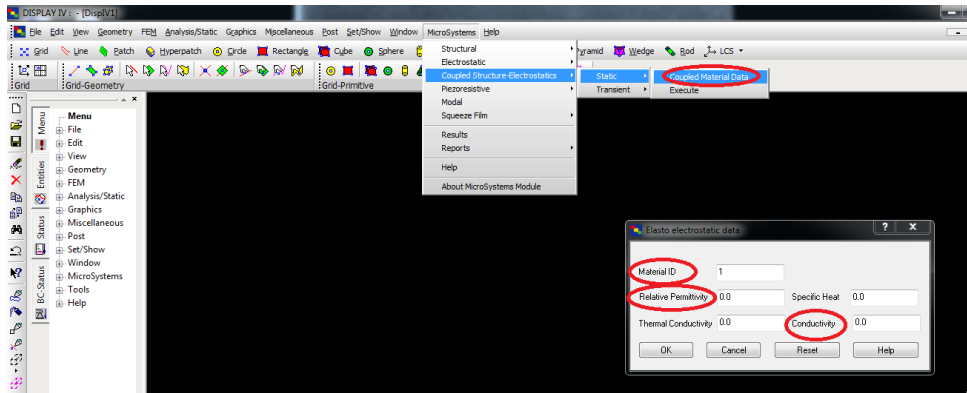

Figure 4.5. Material property data specification from material data section of `coupled structure-electrostatic` analysis.

---

[23]So, there are two locations from where material data has to be specified first, from the `NISA` menu and second, from the coupled analysis section. Do check that you are specifying material data for same material ID from both the locations.

- **Mesh Details**: After material data specification, meshing is to be done for each Hyper patch. Figure (4.6) shows the FEM mesh toolbar in `NISA` used for coupled analysis. For mesh specification, only `FEG` option $\rightarrow$ `Wedge` or `Hexahedron` types are used.[24]
  - In the `Hexahedron` section, Figure (4.7), `3D Solid` is used for conventional displacement based FEM and `3D Hybrid Solid` is used for hybrid FEM. [25] [26]
  - In the case of `Wedge` section, Figure (4.8), `3D Solid` is used for the conventional displacement based FEM and `3D Concrete` is used for hybrid FEM.
  - Most of the parts of microsystems are thin and flat. Use only hybrid elements for meshing such parts.
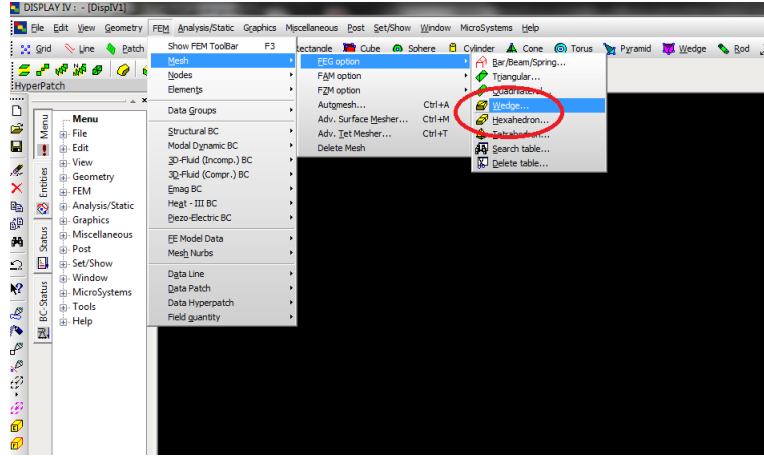

Figure 4.6. Meshing for coupled structure-electrostatic analysis

---

[24]It should be noted that meshing is done on Hyper patch and requirement of nodal continuity must be satisfied on Patch shared by two adjacent Hyper patches. This is the same requirement as mentioned in the meshing section of structural analysis.

[25]Do not use any other element type. Only above mentioned types are used for coupled analysis. Also note that first, one has to select appropriate Hyper patch by clicking `Input Hyp IDs` and then mesh it by specifying number of elements in the `E1/E2/E3` directions.

[26]While meshing each Hyper patch, do not forget specifying associated `Material ID`.
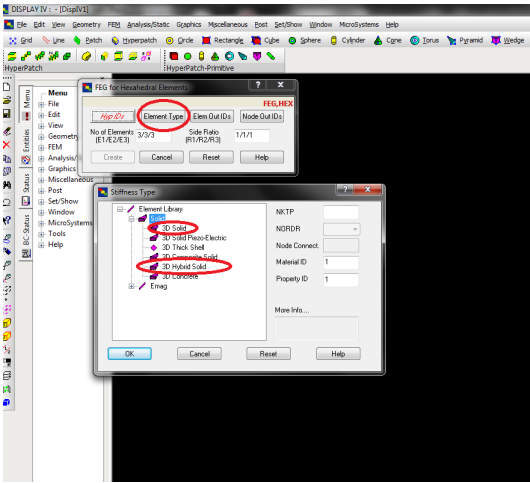
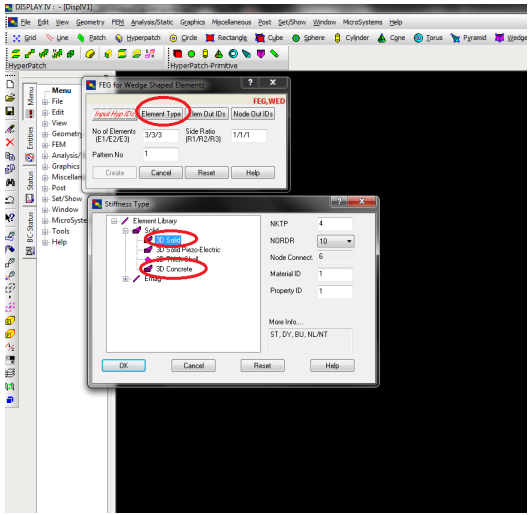**Figure 4.7.** Meshing for coupled analysis: Hexahedron

Figure 4.8. Meshing for coupled analysis: Wedge

- **Structural Boundary Conditions**: There can be four types of boundary conditions specified for coupled structure-electrostatic analysis; [27]
    - Displacement boundary conditions (Figures (4.9) and (4.10))
    - Nodal force boundary conditions (Figures (4.11) and (4.12))
    - Pressure boundary conditions (Figures (4.13) and (4.14))
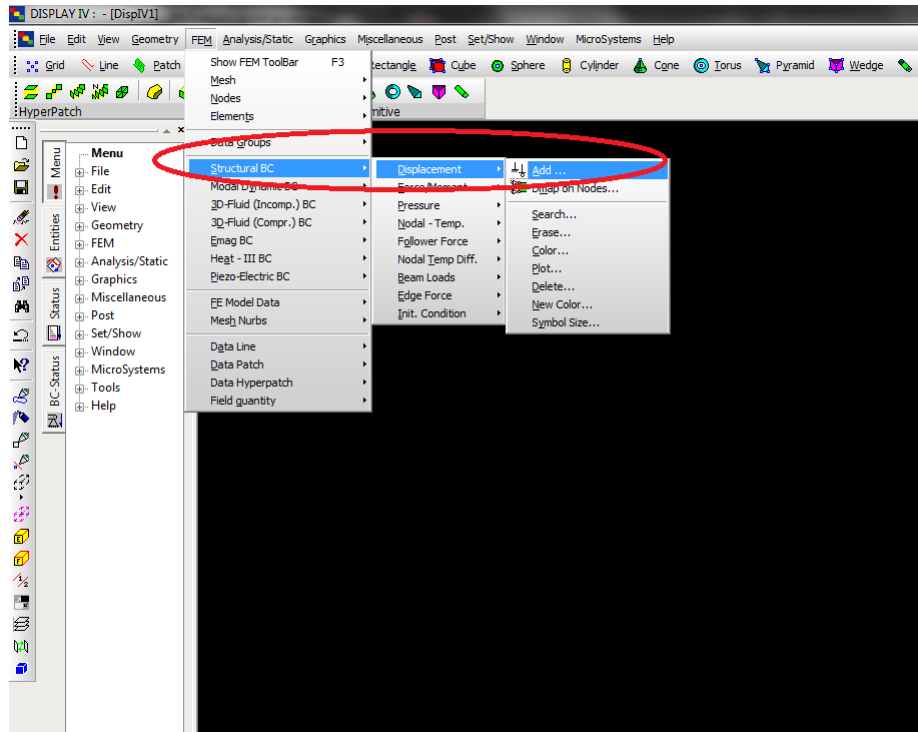   - Applied potential conditions (Figure (4.15))

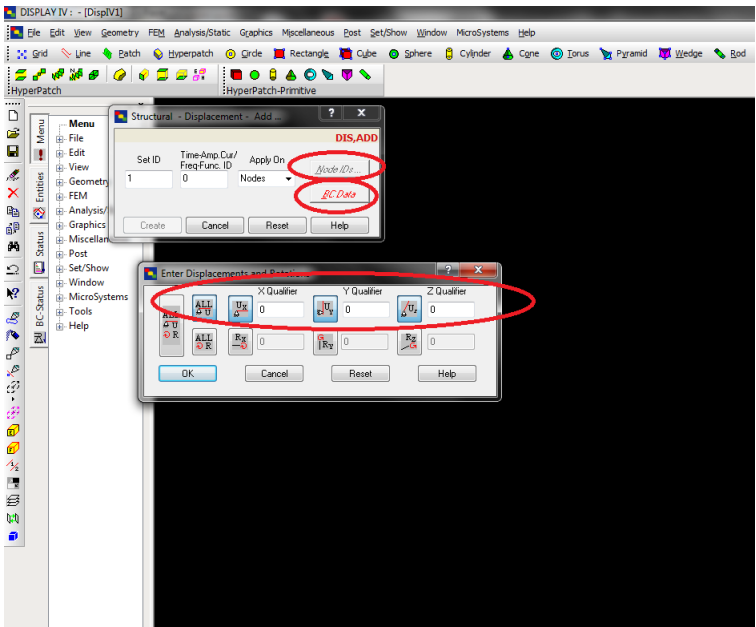**Figure 4.9.** Boundary conditions: Displacement

Figure 4.10. Boundary conditions: Displacement
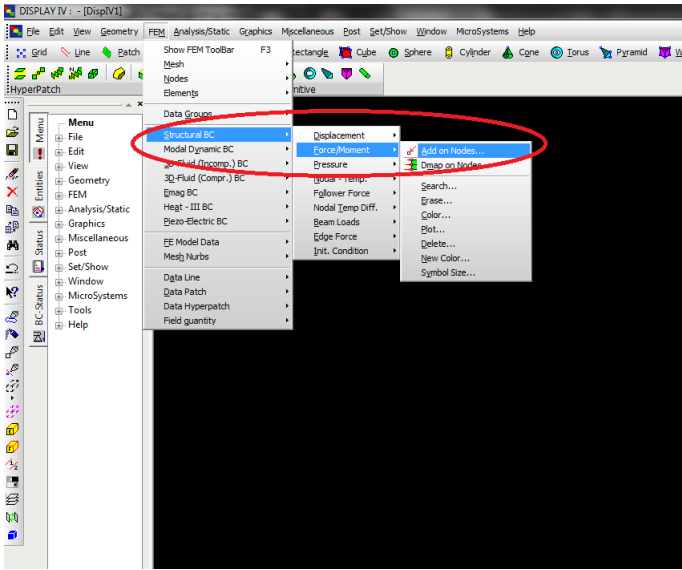

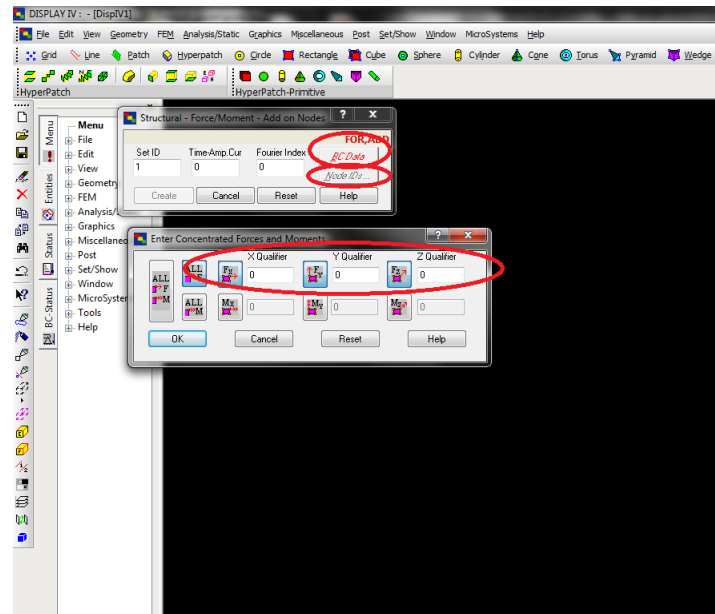**Figure 4.11.** Boundary conditions: Nodal force
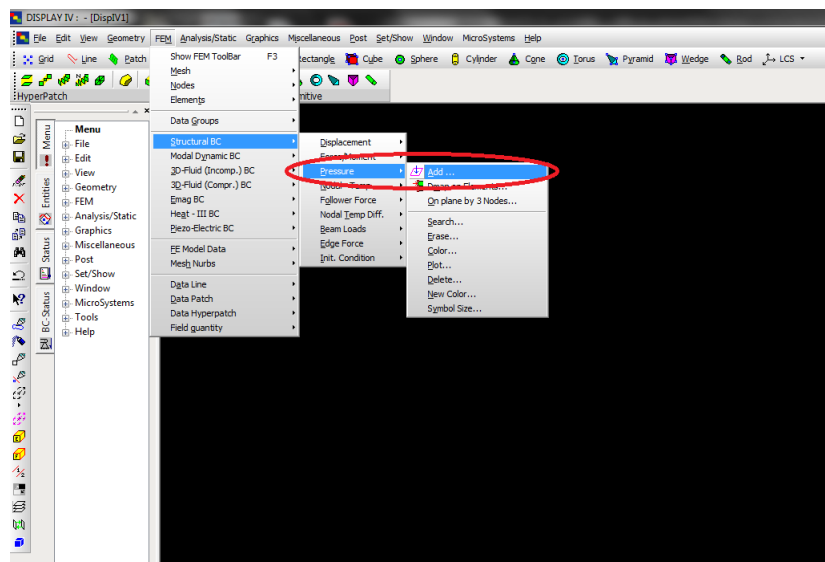
Figure 4.12. Boundary conditions: Nodal force

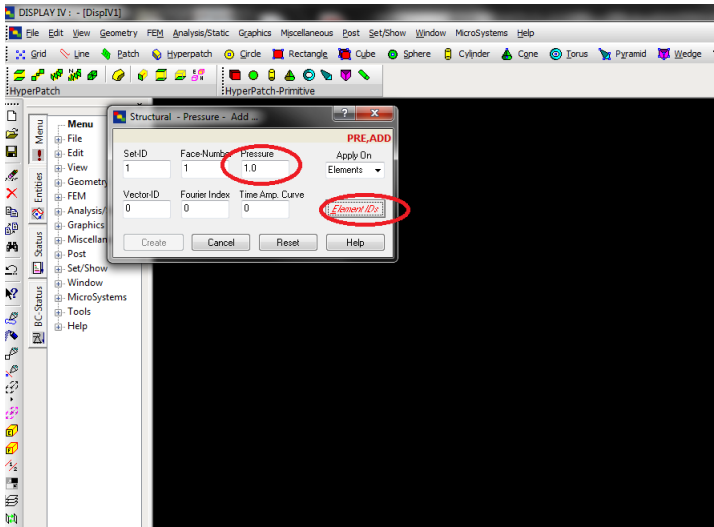**Figure 4.13.** Boundary conditions: Pressure

Figure 4.14. Boundary conditions: Pressure
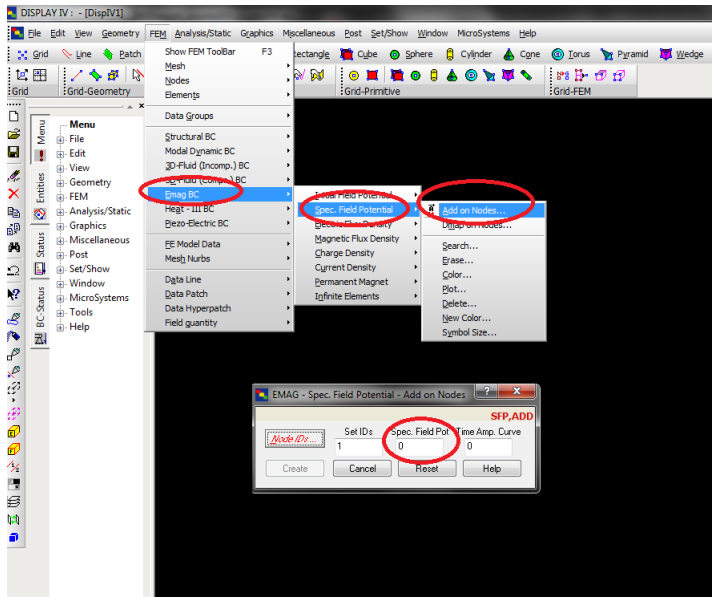

Figure 4.15. Applied potential specifications

4.2 Analysis

4.2.1 Static analysis

Before executing static analysis, save the file as <file>.DBS and <file>.NIS. [28]
Click on the section Static as shown in the Figure (4.16) (if file is not open then
load the <file>.NIS from File menu or load using Run section which is opened
by clicking Execute as shown in Figure (4.16)). Specify the 'node of interest',
at which some useful information (displacement, potential etc.) is required. [29].
Click OK to start the analysis.

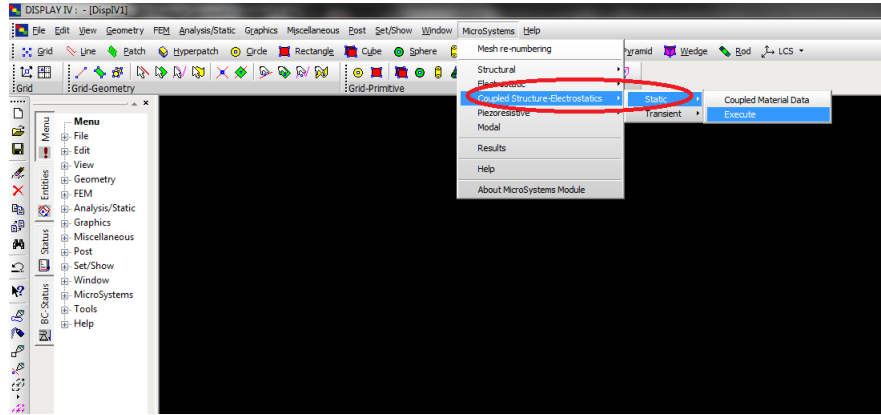Wait for the analysis to get completed. [30].

Figure 4.16. Coupled structure-electrostatic analysis: Static

4.2.2 Transient analysis

Before executing transient analysis, first specify the Time Data as shown in the
Figure (4.17).
Starting Time is 0.0 sec,
End Time is the end of simulation time (in sec) and
Time Step represents the increment in the time (in sec).

Save the file as <file>.DBS and <file>.NIS else if not open, load the <file>.NIS
from File menu or using Run section by clicking Execute section, see Figure
(4.18). Select the 'node of interest' at which information is sought. Click OK.
Wait for the analysis to get completed.

---

[28]The default directory is C:\NISARUN.
[29]This   information   is   used   to   create   some   2D   plots   in   the   directory
C:\NISARUN\Microsystems\results\2dplots
[30]Generally screen gets busy once analysis starts. The end of analysis is indicated by display
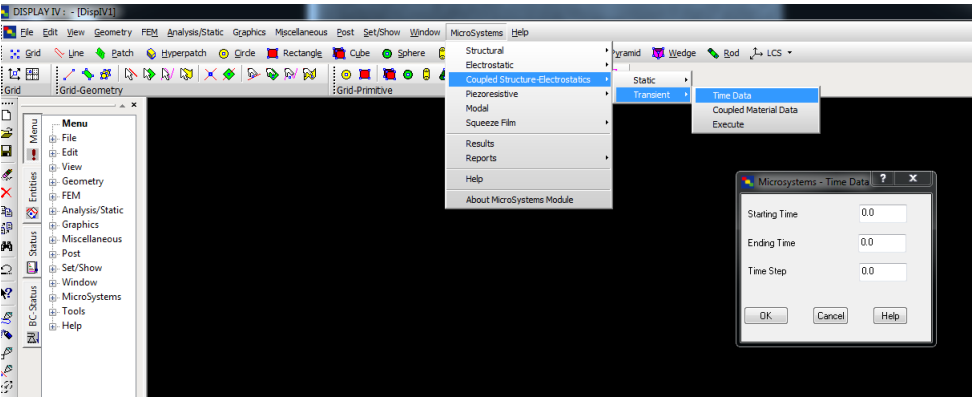of computation times, see Figure (2.17).

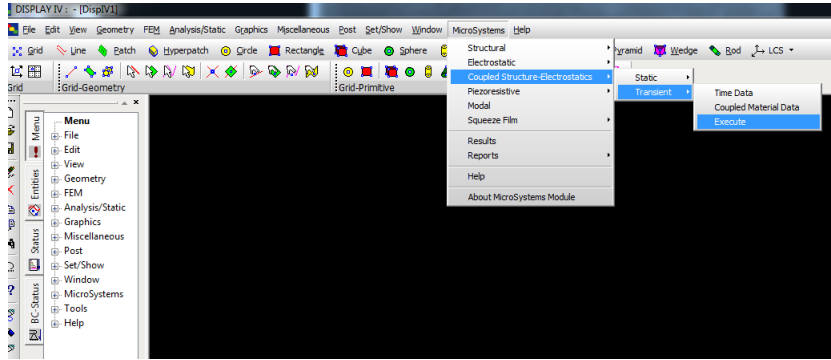**Figure 4.17.** Coupled structure-electrostatic transient analysis time data

Figure 4.18. Coupled structure-electrostatic transient analysis

4.3 Post-processing

After completing the analysis, the output files are written in the directory `C:\NISARUN\Microsystems\results`. The `results` directory has following structure;

- `fields` directory
- `2dplots` directory and
- Squeeze film output files `sfdoutput.dat`, `sfdpostpro.dat` and `sfdpres.dat`.

For visualization, `fields` directory consists of information related to four basic field quantities,

- Displacement (`displacement.dat`)
- Normal stresses (`normal_stress.dat`)
- Shear stresses (`shear_stress.dat`)
- Potential distribution (`potential.dat`)

In order to visualize displacement, load the file `displacement.dat`, from the `Results` section of `Microsystems` module. (see Figure 2.21)

For visualization of normal stresses, load the file `normal_stress.dat`, for shear stresses, load the file `shear_stress.dat` and for potential, load the file `potential.dat` from the `Results` section.

Once appropriate files are loaded from the `Results` section, visualization of field variable is then realized from the menu section as shown in the Figure (2.22).

At present, distribution of field variable over geometry, and maximum and minimum values of field variables with its nodal location can be obtained from GUI.

Animation of field quantities can also be visualized using `Animation` section as shown in the Figure (2.23).

The directory `2dplots`, constitutes following files for coupled analysis;

- `plotlsxx.dat` (factu (factor of load applied) v/s normal stress (x dir))
- `plotlsxy.dat` (factu v/s shear stress (x-y dir))
- `plotlsxz.dat` (factu v/s shear stress (x-z dir))
- `plotlsyy.dat` (factu v/s normal stress (y dir))
- `plotlsyz.dat` (factu v/s shear stress (y-z dir))
- `plotlszz.dat` (factu v/s normal stress (z dir))
- `plotlux.dat` (factu v/s displacement (x dir))
- `plotluy.dat` (factu v/s displacement (y dir))
- `plotluz.dat` (factu v/s displacement (z dir))
- `plotltx.dat` (time v/s displacement (x dir))
- `plotlty.dat` (time v/s displacement (y dir))
- `plotltz.dat` (time v/s displacement (z dir))
- `plotx.dat` (Applied voltage v/s displacement (x dir))
- `ploty.dat` (Applied voltage v/s displacement (y dir))
- `plotz.dat` (Applied voltage v/s displacement (z dir))

These files represents variation of one field quantity with respect to other and can be plotted with `Graph` section as shown in the Figure (2.24).

4.4   Example files

Please see the video's on youtube channel `https://www.youtube.com/user/conelabiisc` (or in the directory coupled analysis in `Microsystems` CD/DVD).

Bibliography

[1] DISPLAYIV User's manual, NISA, Version 17.0.

[2] C. S. Jog and P. P. Kelkar, Non-linear analysis of structures using high performance hybrid elements, *International Journal for Numerical Methods in Engineering*, Volume 68, Issue 4, pages 473-501, 2006.

[3] C. S. Jog and Phani Motamarri, An energy-momentum conserving algorithm for nonlinear transient analysis within the framework on hybrid elements, *Journal of Mechanics of Materials and Structures*, Vol. 4, No. 1, 2009.

[4] C. S. Jog, G.K. Ananthasuresh, M. Sundaram, R. Ganesh, K. Pavan and B. Varun, Static elastic simulation of micromechanical structures using hybrid finite elements, *International Conference on Smart Materials Structures and Systems, Bangalore*, 2012.

[5] C. S. Jog, Kunal D. Patil and G.K. Ananthasuresh, Monolithic hybrid finite element strategy for coupled structure-electrostatic analysis of micromechanical structures, *International Conference on Smart Materials Structures and Systems, Bangalore*, 2014.

[6] C. S. Jog, Kunal D. Patil, G.K. Ananthasuresh and S. Balakrishnan, A simulation module for microsystems using hybrid finite elements: an overview, *Micro and Smart Devices and Systems, Springer Tracts in Mechanical Engineering*, pp 355-373, 2014.

[7] C. S. Jog and Kunal D. Patil, A monolithic hybrid finite element strategy for Micromechanical structures, *manuscript under preparation*