



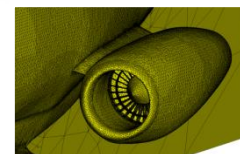
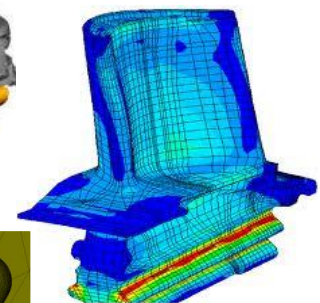
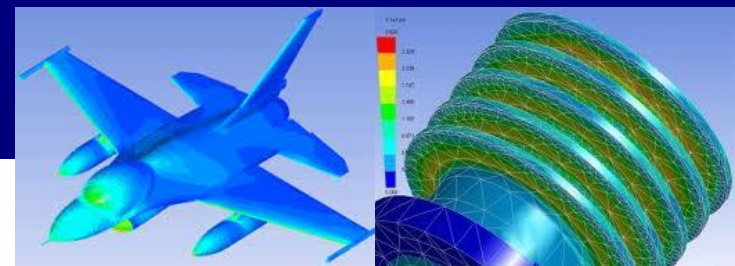
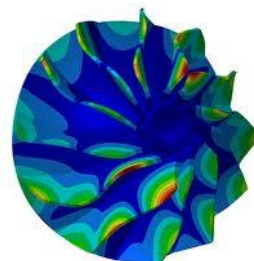
Работа в конечно-элементном пакете

ANSYS[®]

Dr. Anna A. Nasedkina

nasedkina@math.sfedu.ru

Institute of Mathematics,
Mechanics and Computer Science
Southern Federal University

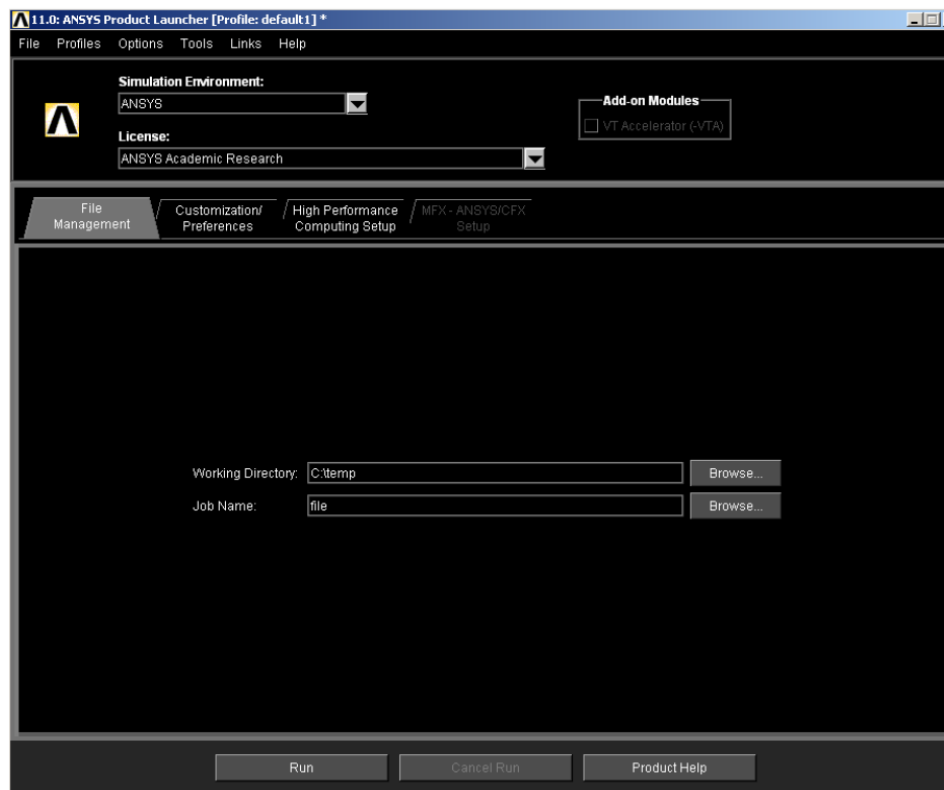


Содержание

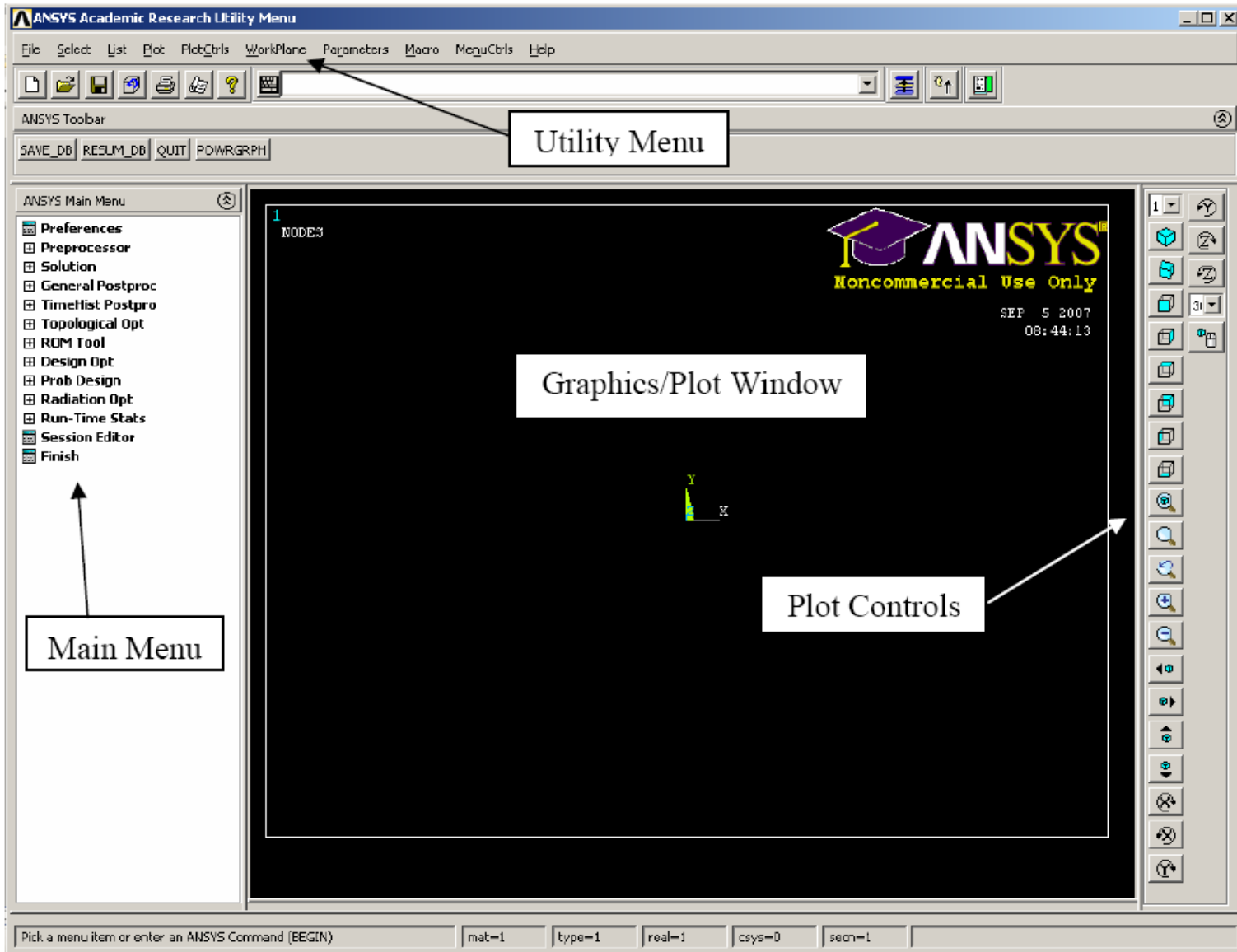
- Запуск Product Launcher
- Графический интерфейс ANSYS (Graphical User Interface)
- Файловая система ANSYS
- Команды и входные файлы (input files)
- Твёрдотельное моделирование (Solid modeling): сверху вниз, снизу вверх, булевы (логические) операции
- Определение типа конечного элемента
- Построение конечно-элементной сетки : свободное разбиение (free mesh), регулярное разбиение (mapped mesh), контроль плотности разбиения
- Задание материальных свойств, использование единиц измерения
- Задание граничных условий
- Пост-процессорная обработка результатов

Запуск Ansys

- Запускаем Product Launcher
- Указываем Simulation Environment, License,
- Задаем рабочую директорию (Working Directory) и имя проекта (Job Name)



Графический интерфейс ANSYS

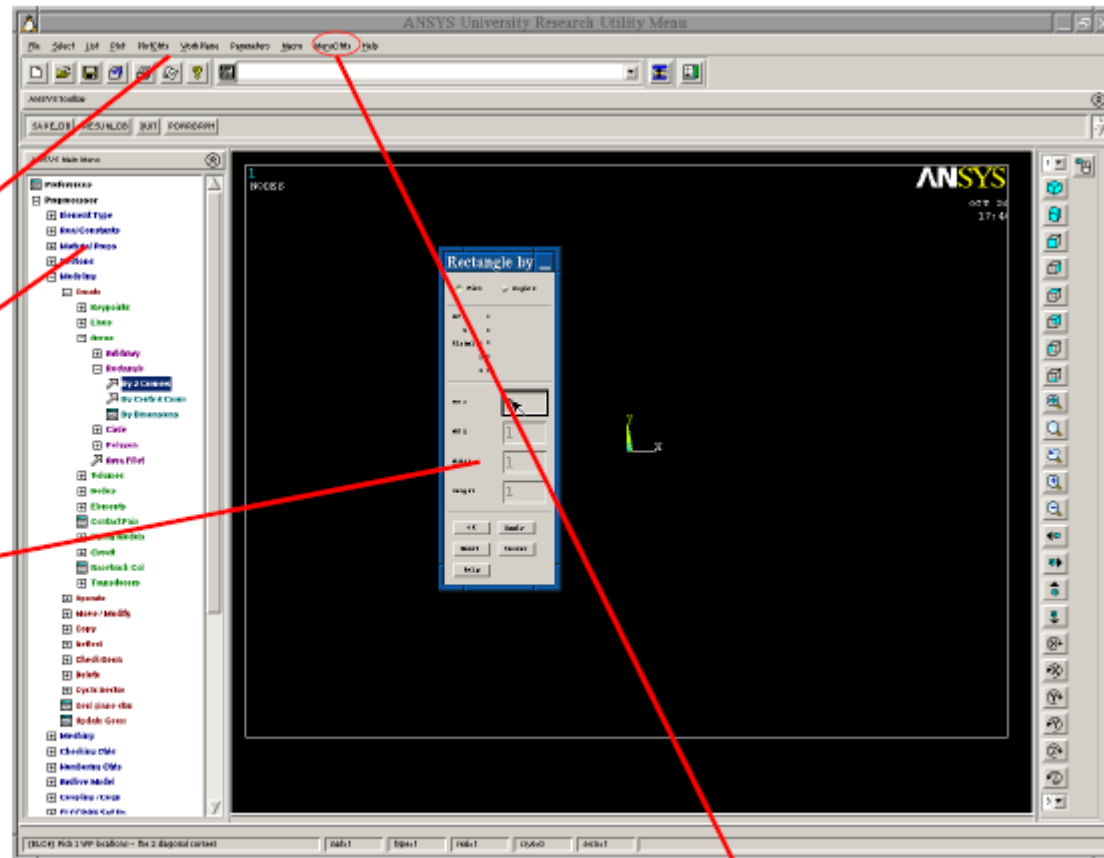


Что можно делать в графическом интерфейсе ANSYS: интерактивный режим

You can do things graphically using the menus

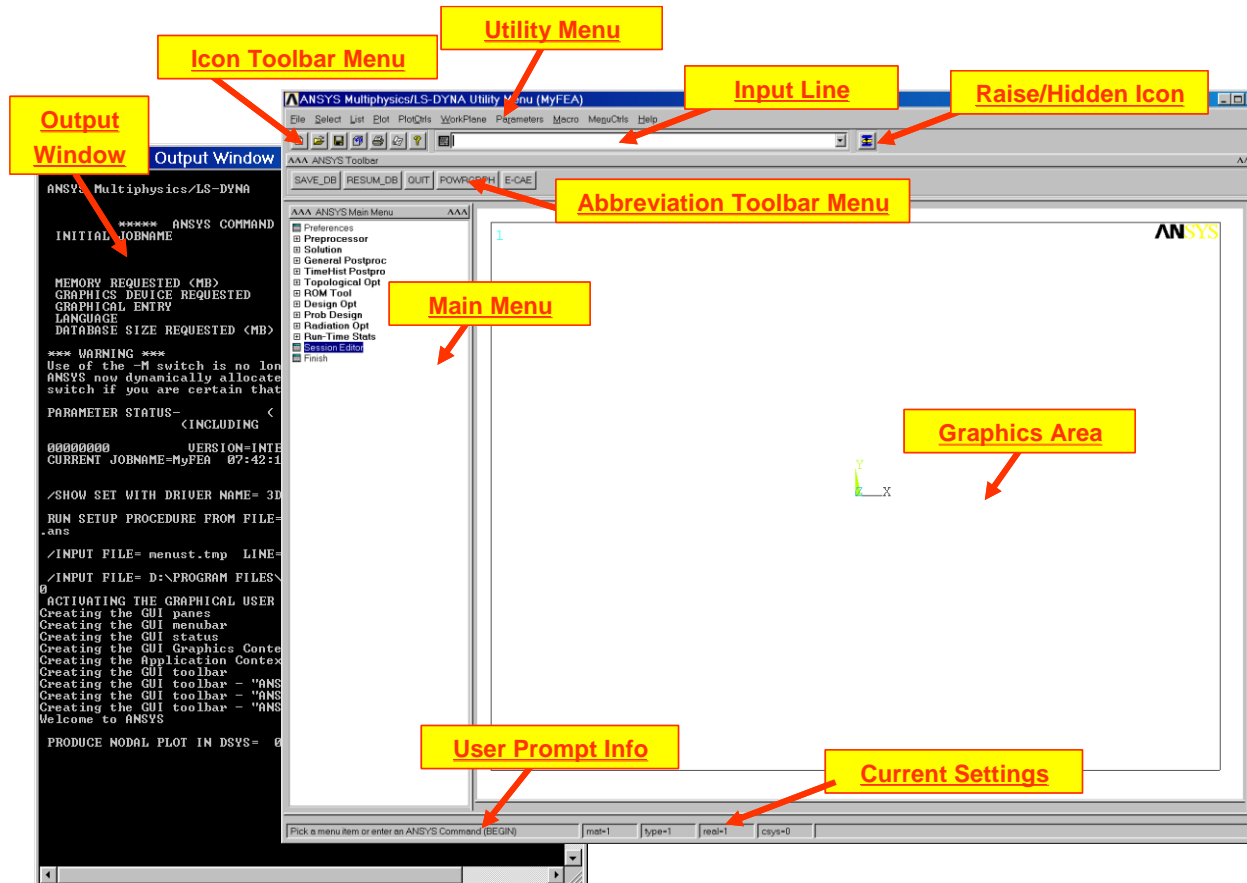
at the top and the side of the screen

This will often bring up a dialog box or ask you to pick things with the cursor



NOTE: You can change font size using the *MenuCtrls > Font Selection* menu

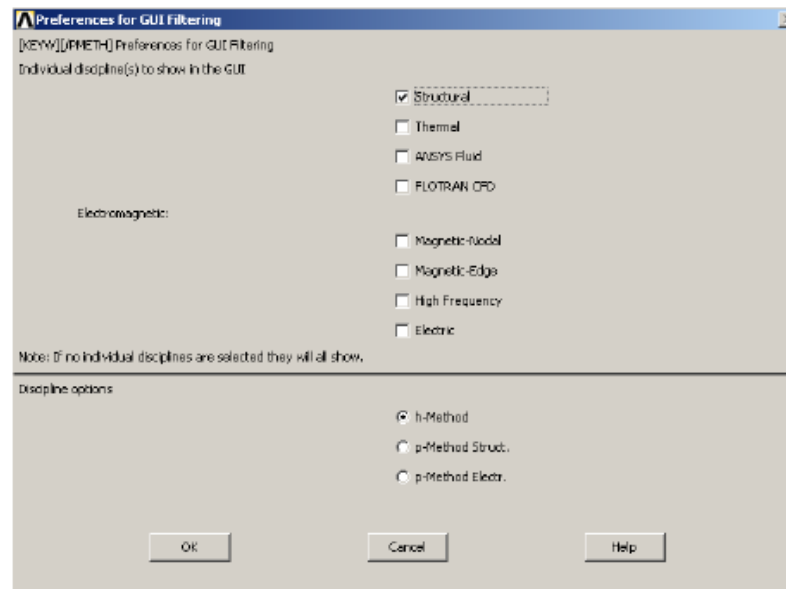
Основные окна ANSYS: graphical user interface и output window



Начальные параметры

Set initial parameters:

- 1) **Utility Menu** → **File** → **Change Jobname...** or **Change Directory...**(In case you forgot when you started ANSYS.)
- 2) **Utility Menu** → **File** → **Change Title...** (This step is not necessary, but it is a good idea. The Title will appear on plots and in other output files.)
- 3) **ANSYS Main Menu** → **Preferences** → Set the **Preferences for GUI Filtering** (This step is not necessary, but it is a good idea. Commands not related to a particular analysis type will be filtered out of the GUI. In this class, we will perform Structural and Thermal analyses. Also, we will generally use the h-method Discipline option.)



Файловая система ANSYS file system

- ❖ **Database and Files:** The term ANSYS database refers to the data ANSYS maintains in memory as you build, solve, and postprocess your model. The database stores both your input data and ANSYS results data:
 - **Input data** -- info you must enter, such as dimensions, material properties, and load data.
 - **Results data** -- quantities that ANSYS calculates, such as displacements and stresses.

❖ Typical files in Ansys

jobname.db, *.dbb*: Database file, binary. Compatible across all supported platforms.

jobname.log: Log file, ASCII. Contains a log of every command issued during the session. If you start a second session with the same jobname in the same working directory, ANSYS will *append* to the previous log file (with a time stamp).

jobname.err: Error file, ASCII. Contains all errors and warnings encountered during the session. ANSYS will also *append* to an existing error file.

jobname.rst, *.rth*, *.rmg*, *.rfl*: Results files, binary. Contains results data calculated by ANSYS during solution. Compatible across all supported platforms.

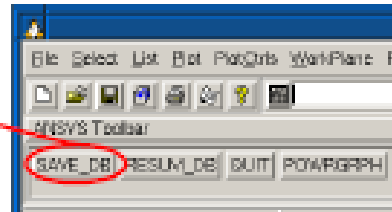
База данных проекта ANSYS

- Сохранение проекта: команда SAVE или кнопка SAVE_DB
- Проект сохраняется в файл с расширением .db
- Для открытия имеющегося проекта нужно выбрать Resume from. Если в Product Launcher уже было задано имя проекта, то выбрать File->Resume или нажать кнопку RESUME_DB

• You can save ANSYS models as a database file

This can be done here

or with the **SAVE** command



- Once you have done this if you list the directory contents it now contains a file: *file.db*
- You can resume from your database each time you start ANSYS

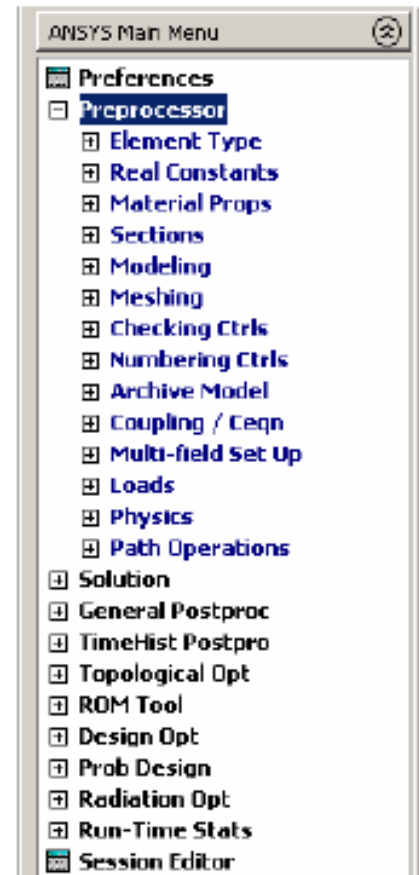
File > Resume from OR **RESUME**

Общая процедура моделирования в ANSYS

- Preprocessor
- Solution
- Postprocessor

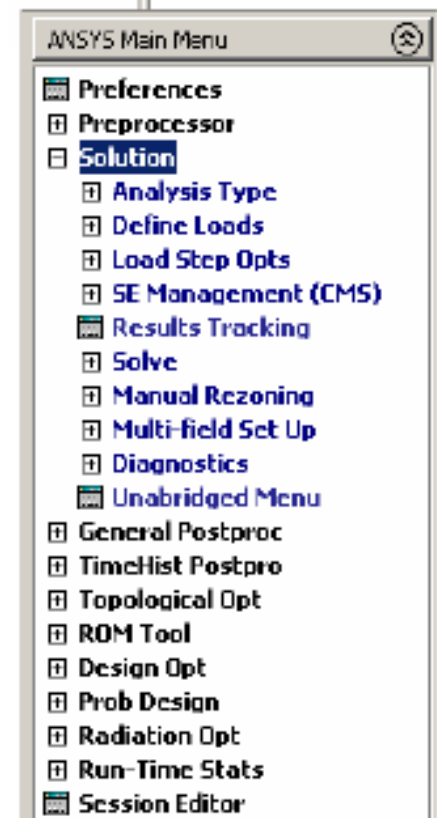
ANSYS Main Menu → Preprocessor

- 1) Define **Element Type**
- 2) Define **Real Constants** (if necessary)
- 3) Define **Material Properties**
- 4) Create the geometric **Model** (define in ANSYS or import geometry from Pro/E or SolidWorks)
- 5) Create a **Mesh** for the model



Общая процедура моделирования в ANSYS

- Preprocessor
- Solution
- Postprocessor

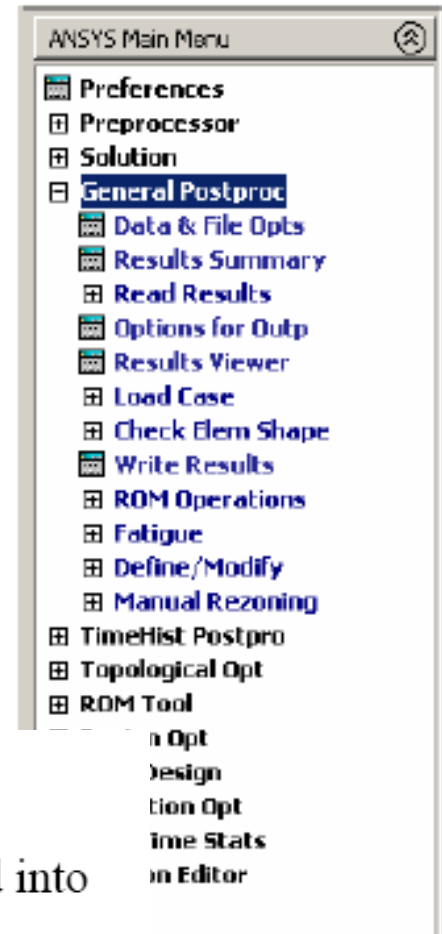


ANSYS Main Menu → Solution

- 1) Define **Analysis Type** (static, harmonic, transient, etc.)
- 2) Define **Loads** (apply loads and constraints/boundary conditions)
- 3) **Solve** the model (run the analysis)

Общая процедура моделирования в ANSYS

- Preprocessor
- Solution
- Postprocessor

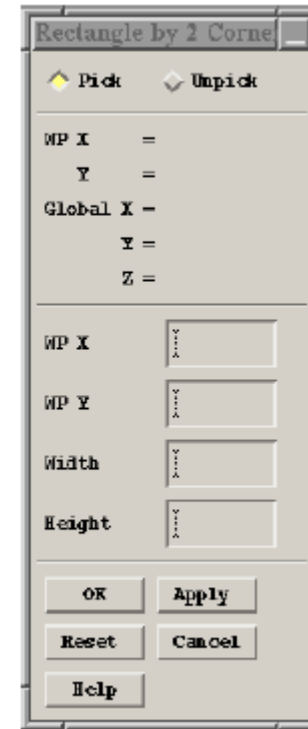


ANSYS Main Menu → General (or TimeHist) Postprocessor

- 1) **Read Results** (extract results from the results file, and read into the database file)
- 2) **Plot** and/or **List** results of interest (e.g., deformed geometry, stresses, strains, displacements, reaction forces, temperatures, etc.)

Графический ввод данных

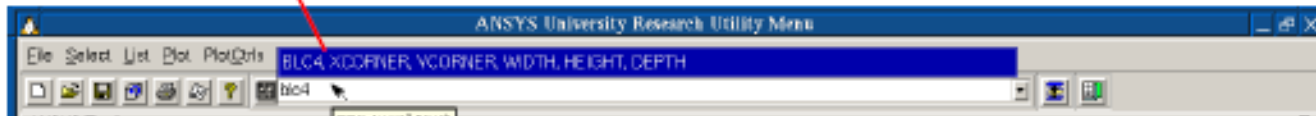
- Try it! From the side menu select:
Preprocessor > Modeling > Create > Areas > Rectangle > By 2 corners
- This should bring up this dialog box
- WP X and WP Y are the coords of the bottom left corner of the rectangle
- Type a value for these and the width and height and click OK
- You should get a cyan rectangle



Использование команд в ANSYS

- Вместо интерактивного режима можно использовать команды
- Команды записываются на встроенном языке программирования APDL – ANSYS Parametric Design Language

- If you type a command in the command box, ANSYS will tell you what it expects as input data



- Typing the command in has the same effect as running it graphically and the command will also be stored in the log file as before
- To get help, simply type *help* in the command box, or use the help menu to select *Help Topics*

Входные файлы (input files) Ansys

- Все этапы моделирования можно записать в виде команд и собрать их в одном входном файле (input file)
- Входной файл – это обычный текстовый файл, он должен иметь расширение .inp или .txt

- You can programme in ANSYS using input files
- These are just text files containing a list of ANSYS commands:

```
!plate: ANSYS input file for plate with hole
/FILNAM,plate ! Name the file
/PREP7 ! Enter the preprocessor
ET,1,PLANE42 ! Define the element type - 4 node solid element
UIMP,1,EX, , ,200000, ! Define material properties
K,1,2.5,0,, ! Define keypoints
LARC,1,2,11,2.5, ! Define arcs
LSTR, 1, 4 ! Define straight lines
AL,3,6,4,1 ! Define areas
lesize,3,, ,15 ! Define mesh size
```

- Purple text is comment (designated by '!') for info only

Преимущества командного режима и работой с входными файлами

- Входные файлы удобны для внесения изменений в расчеты
 - They are small, large models can be built with only a few commands
 - They allow you to review what you've done easily
 - They allow you to modify models very quickly

I have built and meshed a square in ANSYS,
now I want to change the length of the model

Graphically

Either: Modify keypoints and then
remesh the areas of the model
Or: Start again from scratch
File > Clear and start new

With an input file

Change 2 values in input file and run
again:
BLC4, val1 , val2 , val3 , val4
File > Read input from

Использование параметров

- Parameterisation can be very useful

```
/PREP7  
ET,1,42
```

!Define the parameters of my problem

```
xlength = 10  
ylength = 20
```

```
xmesh = 5  
ymesh = 15
```

We can define the mesh and geometry using just these 4 values

!Use the parameters to define geometry and mesh the model

```
BLC4, -xlength/2, -ylength/2, xlength, ylength
```

```
LSEL,s,loc,x,0
```

```
LESIZE,all,,xmesh
```

```
LSEL,s,loc,y,0
```

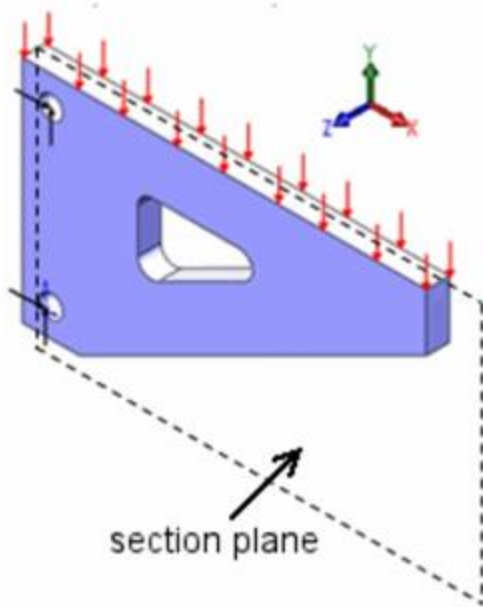
```
LESIZE,all,,ymesh
```

```
AMESH,all
```

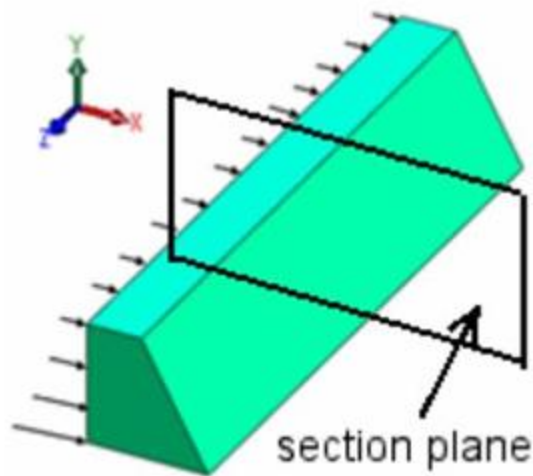
Трехмерное и двумерное моделирование

Переход к двумерной постановке:

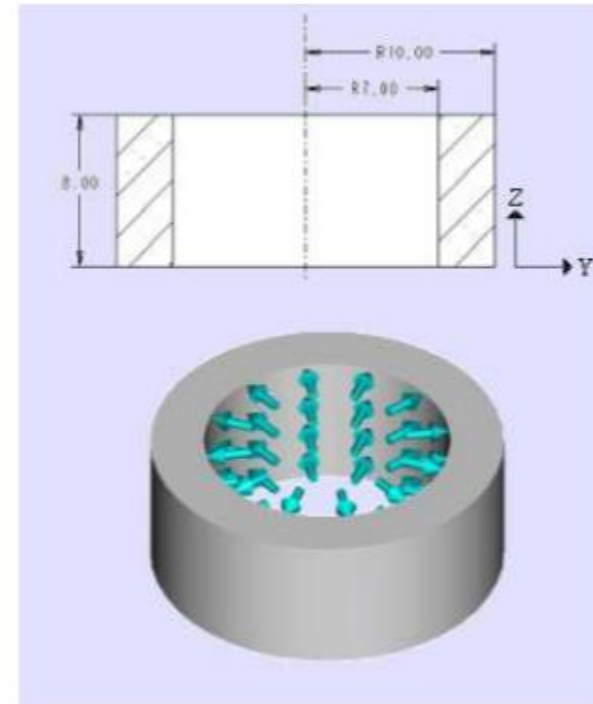
- Плоское напряженное состояние (plane stress)
- Плоская деформация (plane strain)
- Осесимметричность (axial symmetry)



Plane Stress



Plane Strain

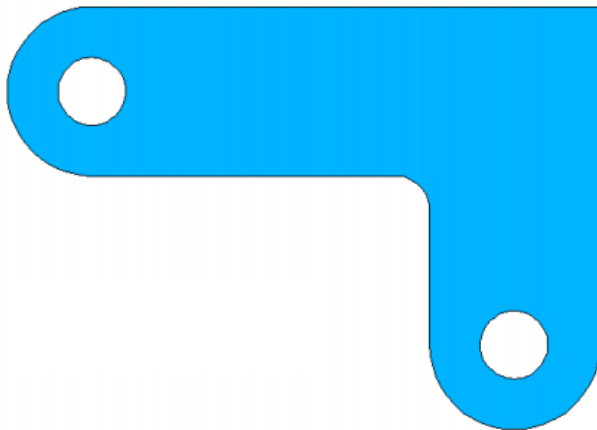


Axisymmetric

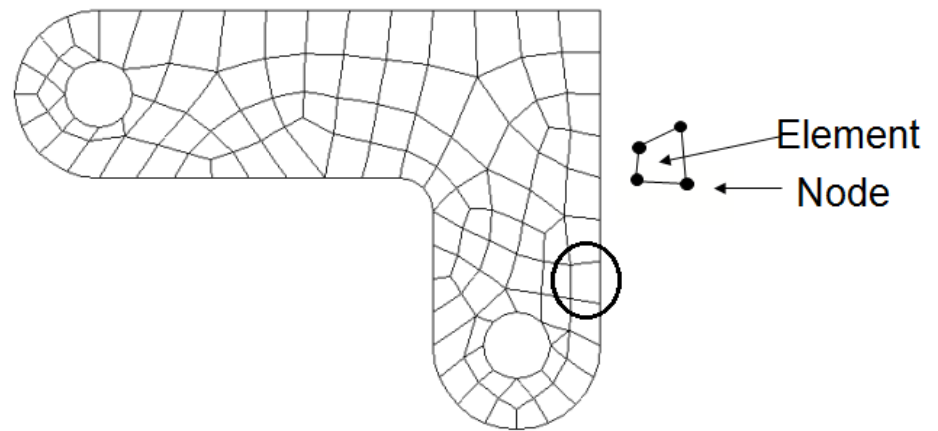
Твердотельное и конечно-элементное моделирование

- Твердотельная модель (solid model)
- Конечно-элементная модель (finite element model)

- Solid model of a bracket



- Finite element model of a bracket



Твердотельное моделирование в ANSYS

❖ Solid Modeling

Be defined as the process of creating solid models in CAD system.

❖ Definitions

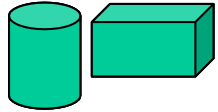
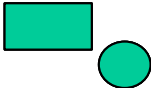


–A *solid model* is defined by volumes, areas, lines, and keypoints.

–Volumes are bounded by areas, areas by lines, and lines by keypoints.

–Hierarchy of entities from low to high as

keypoints < lines < areas < volumes

–You cannot delete an entity if a higher-order entity is attached to it. Also, a model with just areas and below, such as a shell or 2-D plane model, is still considered a solid model in ANSYS terminology.

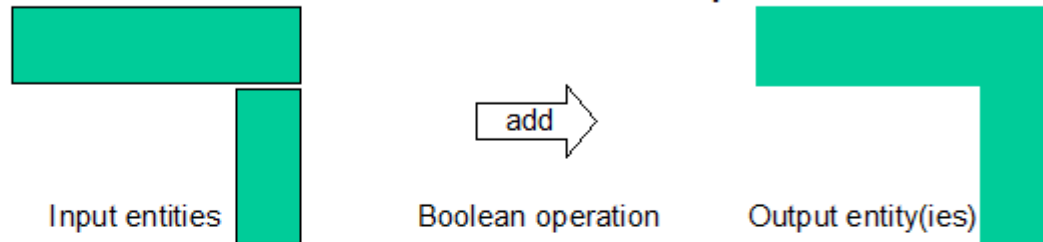
Volumes	
Areas	
Lines	
Keypoints	

Методы твердотельного моделирования (solid modeling) в ANSYS

- Моделирование сверху вниз (top-down)
- Моделирование снизу вверх (bottom-up)

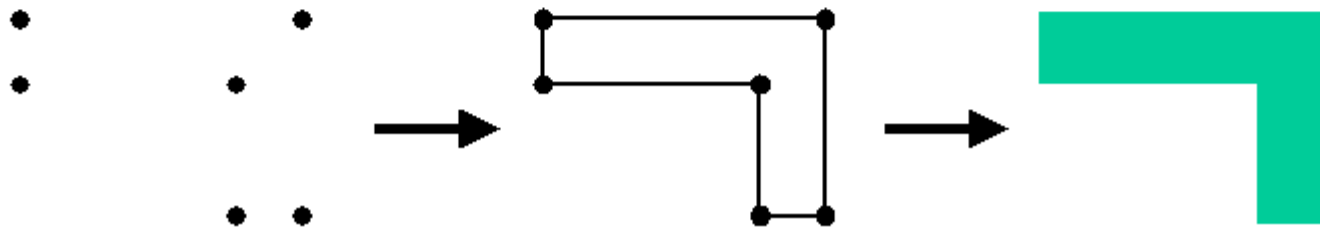
❖ Top-down modeling

starts with a definition of volumes (or areas), which are then combined in some fashion to create the final shape.



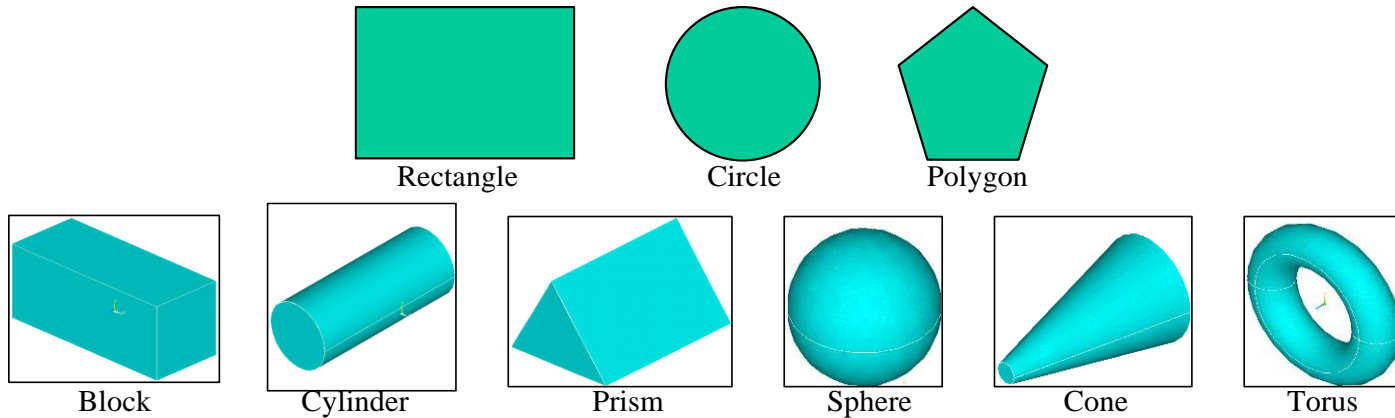
❖ Bottom-up modeling

starts with keypoints, from which you “build up” lines, areas, etc.



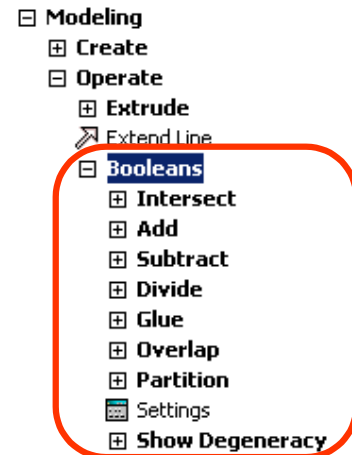
Примитивы ANSYS

The volumes or areas that you initially define are called *primitives*, which are basic entities for the top-down method. ANSYS contains the following 2D and 3D *primitives*:



Булевы (логические операции) при моделировании сверху вниз (top-down method)

The final shape of an object is usually not as simple as primitives. However, it is likely doable to combine a number of primitives through a series of proper *Boolean operations*. The “input” to Boolean operations can be any geometric entity, ranging from simple primitives to complicated volumes generated in previous steps.

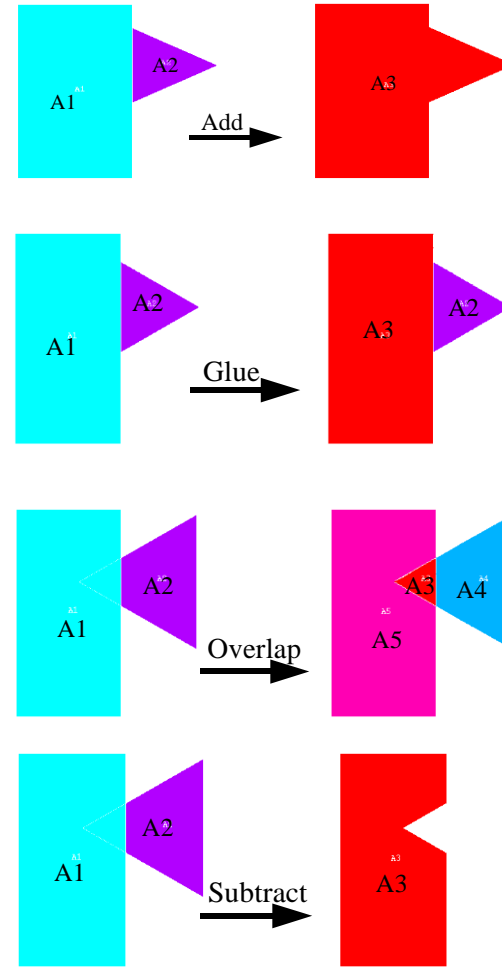


❖ **Boolean operations** are computations involving combinations of geometric entities. ANSYS Boolean operations include add, subtract, intersect, divide, glue, and overlap. • All Boolean operations are available in the GUI:

Main Menu > Preprocessor > Modeling > Operate > Booleans

Некоторые булевы операции в ANSYS

- ❖ **Add:** Combines two or more entities into one.
- ❖ **Glue:** Attaches two or more entities by creating a common boundary between them, which is useful when you want to maintain the distinction between entities (such as for different materials).
- ❖ **Overlap:** Same as glue, except that the input entities overlap each other.
- ❖ **Subtract:** Removes the overlapping portion of one or more entities from a set of “base” entities, which can be useful for creating holes or trimming off portions of an entity.



Задание типа конечного элемента в ANSYS

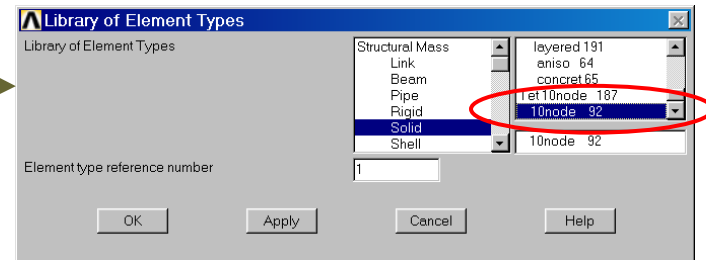
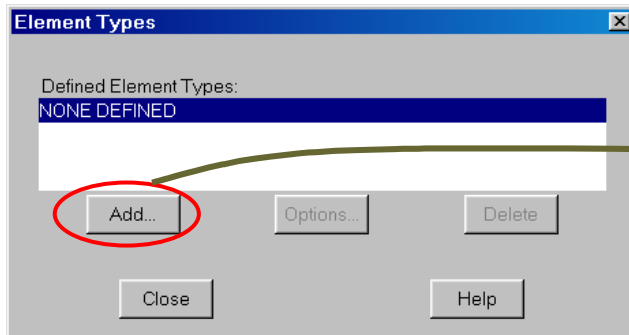
Element Type

The element type is an important choice that determines the following element characteristics:

- Degree of Freedom (DOF) set. A thermal element type, for example, has one dof: TEMP, whereas a structural element type may have up to six dof: UX, UY, UZ, ROTX, ROTY, ROTZ.
- Element shape -- brick, tetrahedron, quadrilateral, triangle, etc.
- Dimensionality -- 2-D solid (X-Y plane only), or 3-D solid.
- Assumed displacement shape -- linear vs. quadratic.

To define an element:

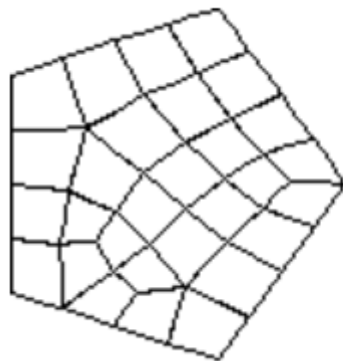
Main Menu>Preprocessor>Element Type> Add/Edit/Delete>Add



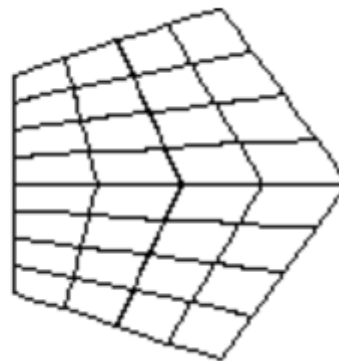
Типы конечно-элементного разбиения: свободное (free) и регулярное (mapped)

- *Свободным* называется разбиение, которое не имеет ограничений на форму элементов и не имеет заданного шаблона сетки. Обычно свободное разбиение используется для областей и объемов сложной формы. Для свободного разбиения достаточно задать средний размер конечного элемента.
- *Регулярное разбиение* определяется заданной формой конечных элементов (например, только треугольники или только четырехугольники в случае разбиения областей, только гексаэдры в случае разбиения объемов) и заданным шаблоном конечно-элементной сетки с четкими «рядами» элементов.

Free mesh



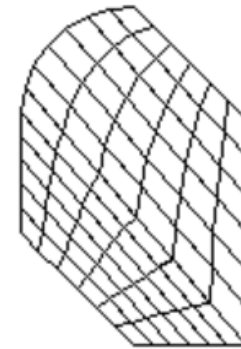
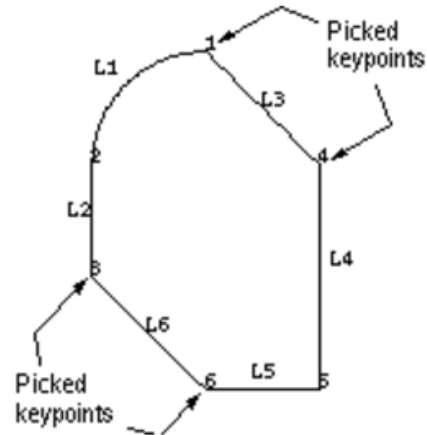
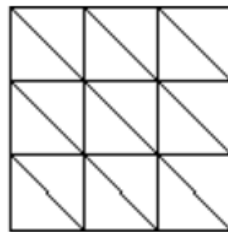
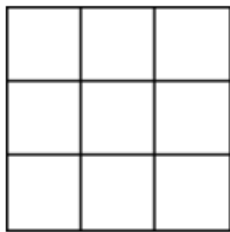
Mapped mesh



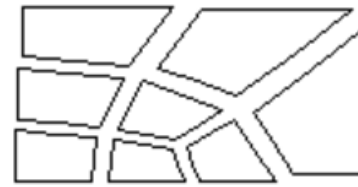
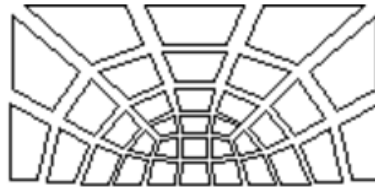
Регулярное разбиение областей

Simplified mapped meshing (AMAP)

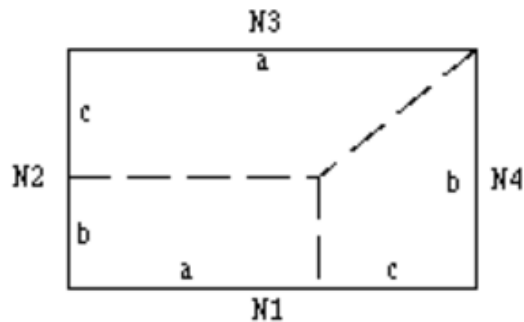
Area mapped meshes



Examples of transition mapped quadrilateral meshes



Регулярное разбиение областей: допустимые шаблоны transition mapped mesh



$$N2-N4 = N1-N3$$

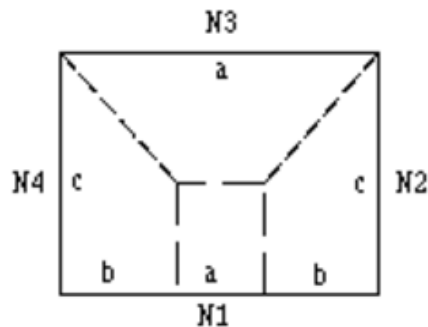
Side 1: has $N1$ divisions where $N1 = a + c$
 Side 2: has $N2$ divisions where $N2 = b + c$
 Side 3: has $N3$ divisions where $N3 = a$
 Side 4: has $N4$ divisions where $N4 = b$

(that is, $N2-N4 = N1-N3$)

For example:

$$\begin{aligned} a &= 3 \\ b &= 2 \\ c &= 3 \end{aligned}$$

$$\begin{aligned} N1 &= 6 \\ N2 &= 5 \\ N3 &= 3 \\ N4 &= 2 \end{aligned}$$



$N2=N4$ and the difference between $N1$ and $N3$ is even

Side 1: has $N1$ divisions where $N1 = a + 2b$
 Side 2: has $N2$ divisions where $N2 = c$
 Side 3: has $N3$ divisions where $N3 = a$
 Side 4: has $N4$ divisions where $N4 = c$

(that is, $N2=N4$ and the difference between $N1$ and $N3$ is even)

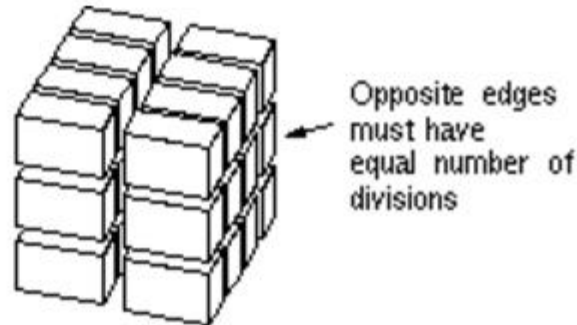
For example:

$$\begin{aligned} a &= 3 \\ b &= 4 \\ c &= 2 \end{aligned}$$

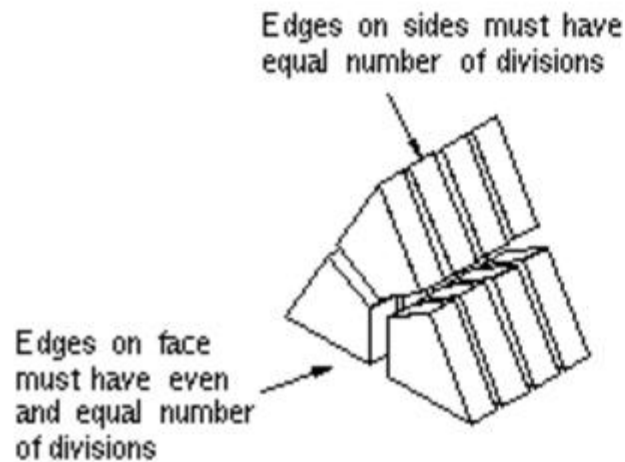
$$\begin{aligned} N1 &= 11 \\ N2 &= 2 \\ N3 &= 3 \\ N4 &= 2 \end{aligned}$$



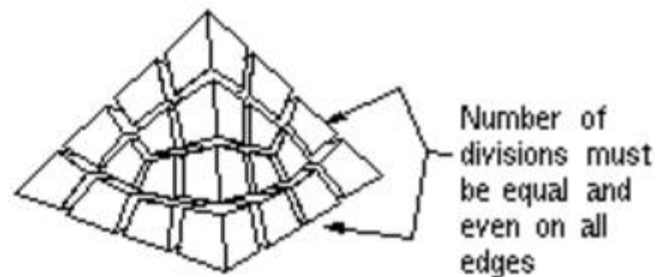
Регулярное разбиение объемов



(a) Brick Shaped Volume

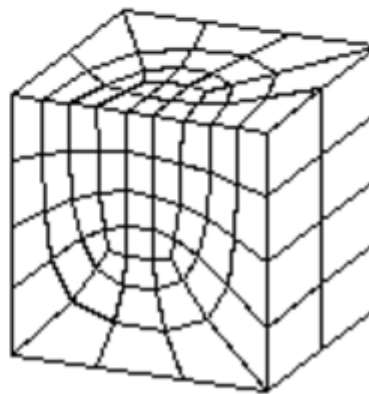
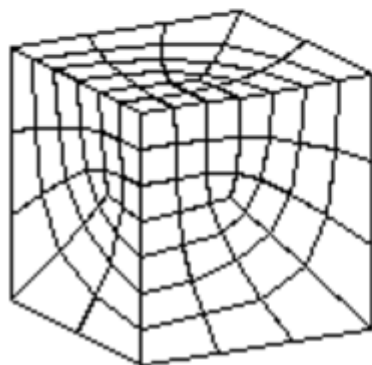
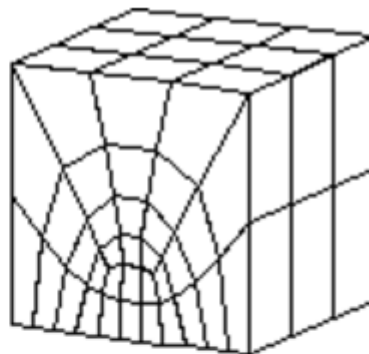
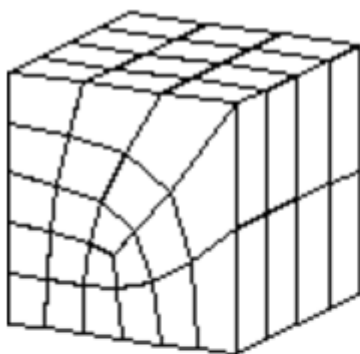


(b) Prism Shaped Volume

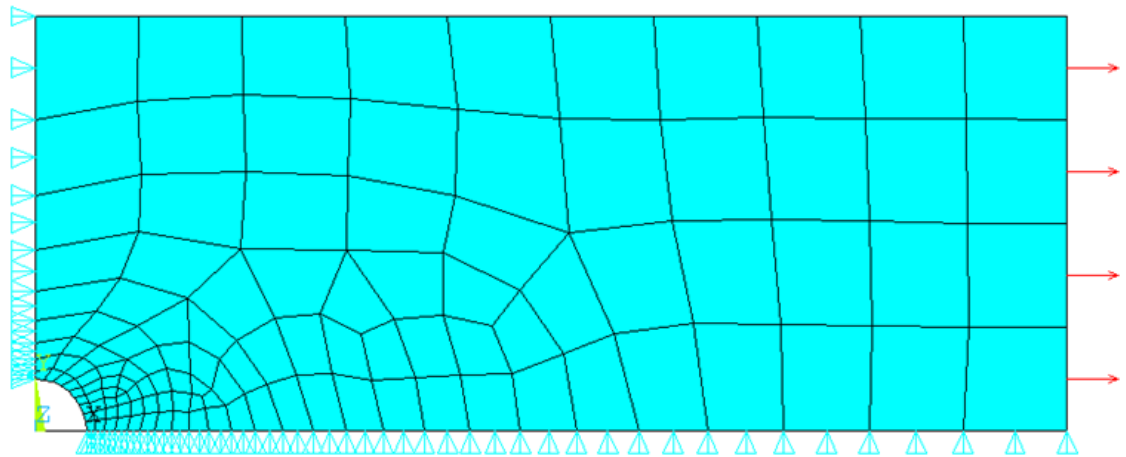
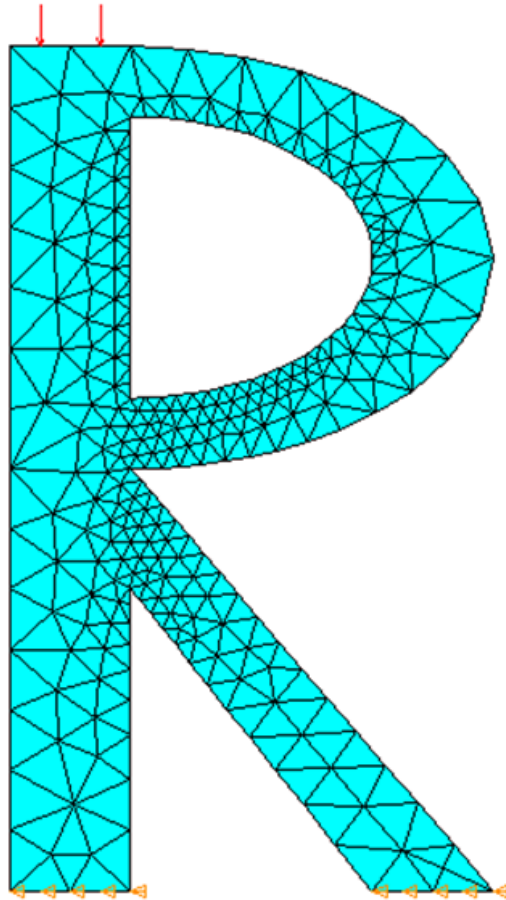


(c) Tetrahedral Shaped Volume

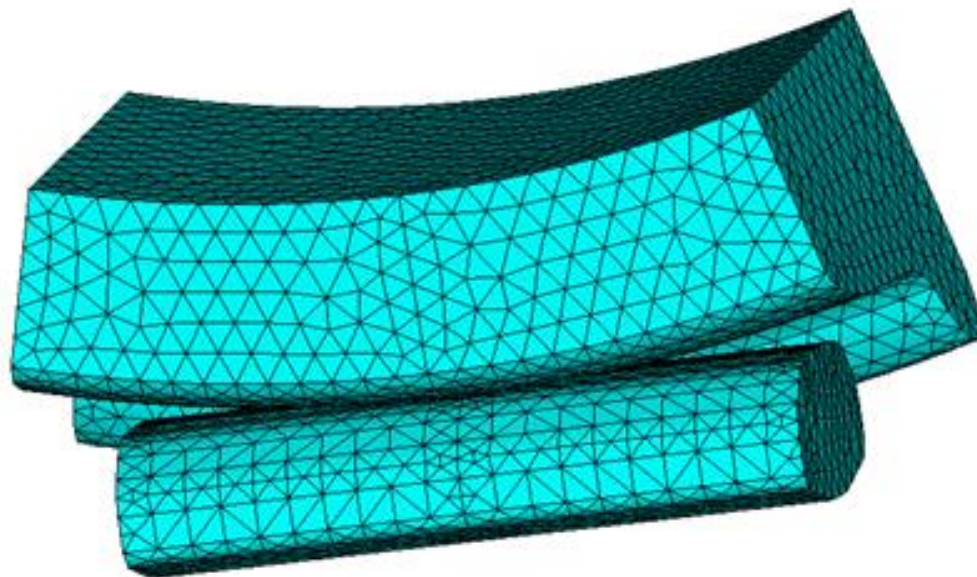
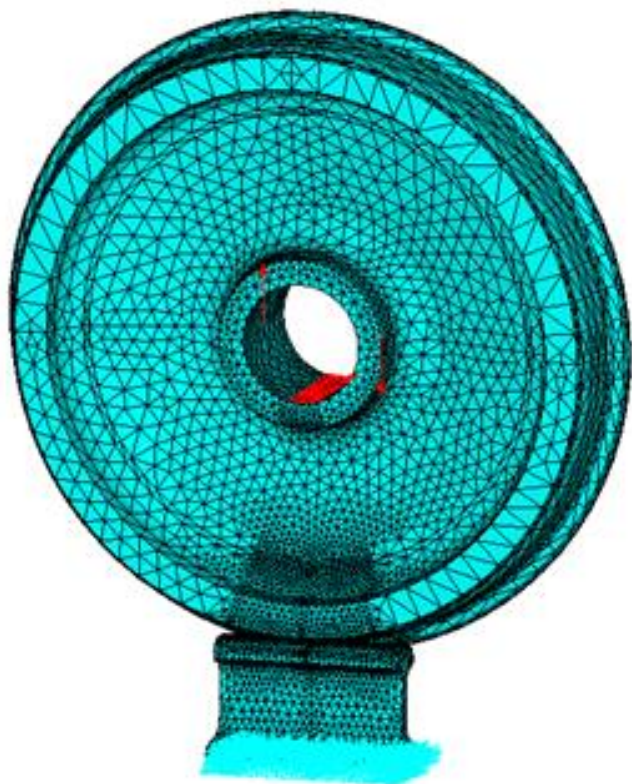
Регулярное разбиение объемов: transition mapped mesh



Примеры свободного разбиения областей



Примеры свободного разбиения объемов



Контроль плотность конечно-элементного разбиения (Mesh density control)

❖ Mesh Density Control

ANSYS provides many tools to control mesh density, on a global and local level:

–Global controls: SmartSizing; Global element sizing; Default sizing

–Local controls: Keypoint sizing; Line sizing; Area sizing

To bring up the MeshTool:

Main Menu > Preprocessor > Meshing > MeshTool

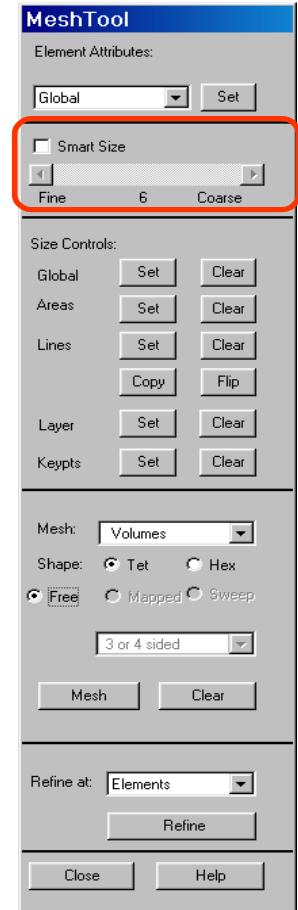
❖ **SmartSizing:** by turning on SmartSizing, and set the desired size level. Size level ranges from 1 (fine) to 10 (coarse), defaults to 6. Then mesh all volumes (or all areas) at once, rather than one-by-one. Advanced SmartSize controls, such as mesh expansion and transition factors, are available by

Main Menu>Preprocessor>Meshing>Size Cntrls>SmartSize>Adv Opts

❖ **Global Element Sizing:** Allows you to specify a maximum element edge length for the entire model (or number of divisions per line):

Go to “Size Controls”, “Global” ,and click [Set] or

Main Menu>Preprocessor>Meshing>Size Cntrls > ManualSize >Global >Size



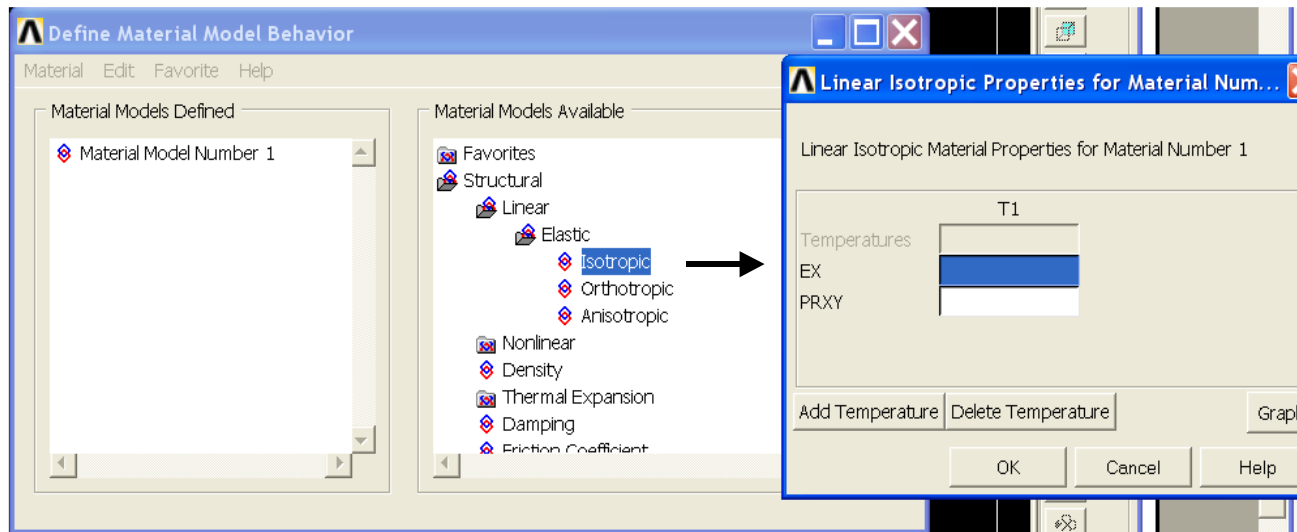
Материальные свойства и единицы измерения в ANSYS

Material Properties

Every analysis requires *some* material property input: Young's modulus (EX), Poisson's ratio (PRXY) for structural elements, thermal conductivity (KXX) for thermal elements, etc.

To define the material properties:

Main Menu>Preprocessor>Material Props>Material Models

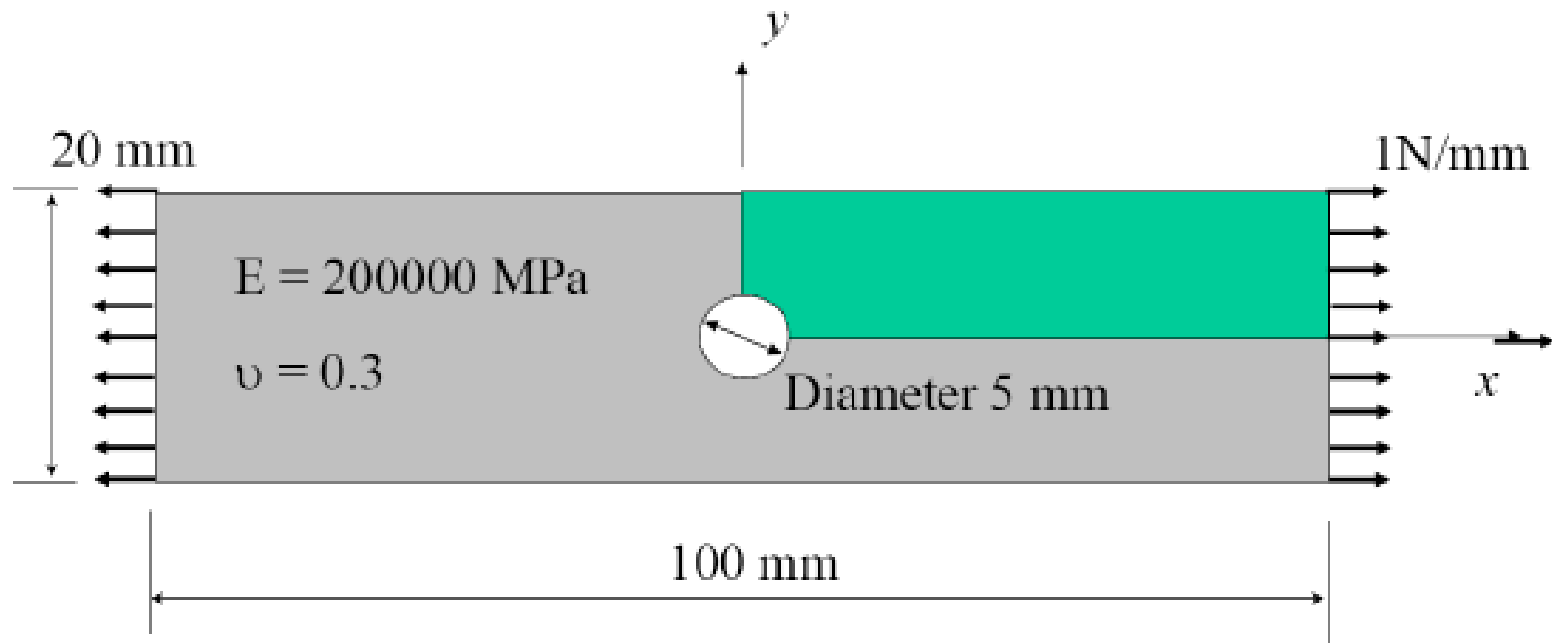


More than one set of material properties can be defined when needed.

❖ **Unit** – The ANSYS program does not assume a system of units for your analysis. You can use any system of units

Пример: задача структурного анализа

- Задача о растяжении пластины. Найти максимальные напряжения в пластине.
- Двумерный случай (плоское напряженное состояние)
- Использование симметрии задачи: $\frac{1}{4}$ пластины



Stress concentration at hole in plate ?

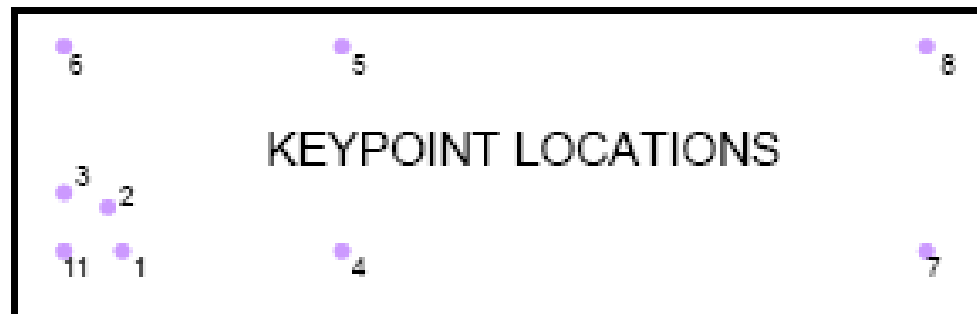
Использование единиц измерения в ANSYS

- Важно: в среде ANSYS Mechanical APDL (в отличие от ANSYS Workbench) единицы измерения не задаются
- Пользователь должен сам следить за соответствием единиц измерения!
 - ANSYS knows nothing about the units you are using
 - It is important to be consistent
 - For this example we will choose **mm** as our unit of length and **MPa** for stress
 - The consistent units of mass, time and force are then **grams**, **ms** and **N** respectively

Пример входного файла для модельной задачи: генерация опорных точек (keypoints)

!plate: ANSYS input file for plate with hole

```
/FILNAM,plate      ! Sets the filename – all output files will be named plate from now on
/PREP7            ! Enter the preprocessor – all geometry creation is done here
ET,1,PLANE42     ! Define the element type - 4 node solid element, type help 42 for more info
UIMP,1,EX,,200000 ! Define material properties, this is the Young's modulus in the x direction
UIMP,1,NUXY,,0.3  ! Define material properties, this is the value of Poisson's ratio for the material
K,1,2.5,0,,      ! Define keypoints, format is: k,knum,kx,ky knum is the number of the point
K,2,1.768,1.768,, ! knum is the number of the point, kx is the x coordinate, ky is the y coordinate
K,3,0,2.5,,
K,4,10,0,,
K,5,10,10,,
K,6,0,10,,
K,7,50,0,,
K,8,50,10,,
K,11,0,0,,
```



Пример входного файла для модельной задачи: генерация линий, областей и конечно- элементной сетки (lines, areas and FE mesh)

```

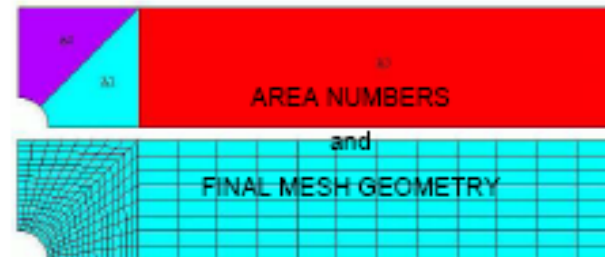
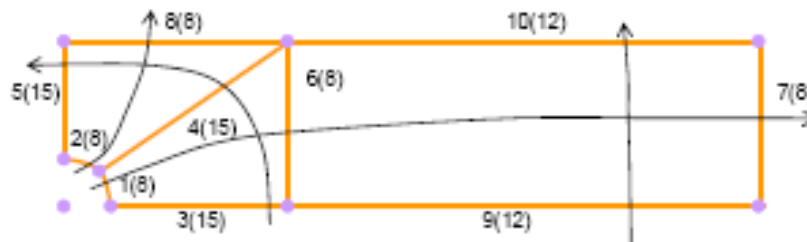
LARC,1,2,11,2.5,    ! Define arcs
LARC,2,3,11,2.5,    ! format is: larc, pt1,pt2,cntr,radius
LSTR, 1, 4          ! Define straight lines between kypts
LSTR, 2, 5
LSTR, 3, 6
LSTR, 4, 5
LSTR, 7, 8
LSTR, 5, 6
LSTR, 4, 7
LSTR, 5, 8
AL,3,6,4,1          ! Define areas using line numbers
AL,4,8,5,2
AL,9,7,10,6
    
```

```

lesize,3,,,15      ! Define mesh size
lesize,4,,,15      ! The number of elements
lesize,5,,,15      ! is specified per line
lesize,1,,,8
lesize,2,,,8
lesize,6,,,8
lesize,7,,,8
lesize,8,,,8
lesize,9,,,12
lesize,10,,,12
amesh,1,3,1         !Mesh Areas
! Then define boundary conditions, loads
! Solve the model and post-process
    
```

LINE NUMBERS AND (MESH SIZING)

For 'mapped meshing' we need consistent sizing in the direction of the arrows – this can reduce the number of elements and generally creates a better mesh



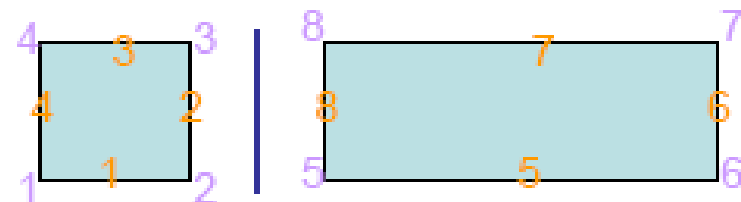
Слияние общих границ при использовании геометрических примитивов

- If you define areas and lines using BLC4 and other commands like it, they will not share boundaries
- You can use **NUMMRG,all,tol/** to merge common boundaries (tol is the tolerance, usually use very small value)

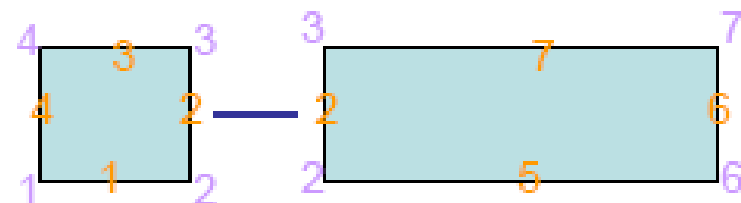
Definition of model geometry



Lines and *kypts* before NUMMRG



Lines and *kypts* after NUMMRG



Задание граничных условий: симметрия (symmetry)

GRAPHICAL METHOD

Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > Choose
and then pick the nodes (or lines) you want using the cursor

COMMAND METHOD

Select nodes by location using **NSEL,s,loc,x,min,max**
where *x,y* or *z* is coordinate direction, *min* is the minimum value in this direction, *max* is the maximum value (if only one value is given, just the nodes at this exact location are selected).
Apply DOF constraints to selected nodes using **D,nnum,DOF,val**
where *nnum* is the number of the node to apply the constraint to (use all for all selected nodes), *DOF* is the label of the DOF (in this case either *UX* or *UY*) and *val* is the value of the DOF.
e.g. **NSEL,s,loc,x,0**
D,all,ux,0

To display the DOF constraints on your model use the menu:

PlotCtrls > Symbols

and select the 'ALL APPLIED BCs' option

Задание граничных условий: сосредоточенная сила (force)

GRAPHICAL METHOD

Preprocessor > Loads > Define Loads > Apply > Structural > Force/Moment > *Choose*
and then pick the nodes (or lines) you want using the cursor

COMMAND METHOD

Select nodes by location using **NSEL,s,loc,x,min,max**

where x,y or z is coordinate direction, min is the minimum value in this direction, max is the maximum value (if only one value is given, just the nodes at this exact location are selected).

Apply forces to selected nodes using **F,num,LAB,val**

where num is the number of the node to apply the force to (use all for all selected nodes), LAB is the label of the force (in this case either FX or FY) and val is the value of the force.

e.g. **NSEL,s,loc,x,50**

F,all,fx,??

Задание граничных условий: давление (pressure)

- Давление – это распределенная нагрузка (Surface Load)

GRAPHICAL METHOD

Preprocessor > Loads > Define Loads > Apply > Structural > Pressure > *Choose*
and then pick the nodes (or lines) you want using the cursor

COMMAND METHOD

Select nodes by location using **NSEL,s,loc,x,min,max**
where x,y or z is coordinate direction, min is the minimum value in this direction, max is the maximum value (if only one value is given, just the nodes at this exact location are selected).

Apply pressures to selected nodes using **SF,nnum,PRES,val**
where nnum is the number of the node to apply the pressure to (use all for all selected nodes), PRES tells ANSYS to apply a pressure and val is the value of the pressure.

e.g. **NSEL,s,loc,x,50**
SF,all,PRES,1*zthick

Конечные элементы и их опции

- 2d structural element, PLANE 42

The nodal forces, if any, should be input per unit of depth for a plane analysis (except for KEYOPT(3) = 3)

KEYOPT(3) defines element behavior:

0 = Plane stress (default) (stress in z direction = 0)

1 = Axisymmetric

2 = Plane strain (strain in z direction = 0)

3 = Plane stress with thickness input

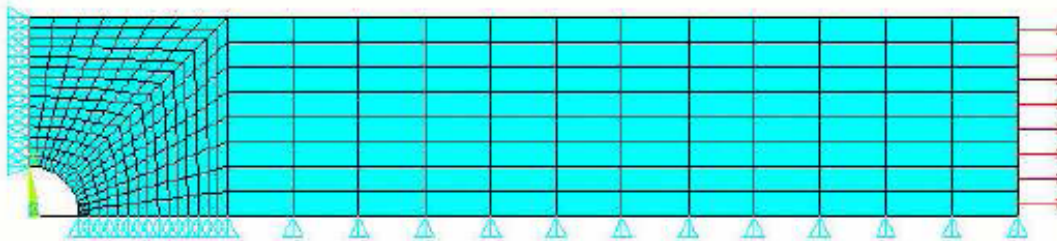
We want to calculate the plane stress solution, forces are per unit of depth in z direction so the effective $zthick = 1$. So we should apply a pressure of $1*1 = 1$.

We can set a different thickness value by setting KEYOPT,ITYPE,3,3 (ITYPE is the element type number = 1) and R, ISET, THK (ISET is the real constant set number = 1 and THK is the thickness value).

Positive pressures act into the element and we want to apply the load acting out, therefore we apply a pressure of -1.

Конечно-элементная модель с граничными условиями; запуск решателя

- What should the loaded model look like?



- So now we can solve the problem

GRAPHICAL METHOD

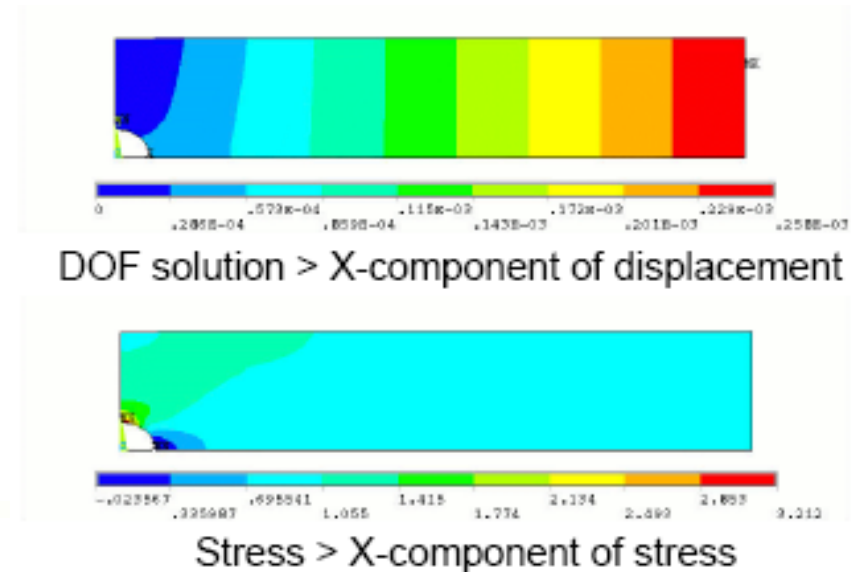
```
Select > Everything  
Solution > Solve > Current LS
```

COMMAND METHOD

```
allsel  
/solu  
solve
```

Просмотр и листинг результатов в общем постпроцессоре (General Postprocessor)

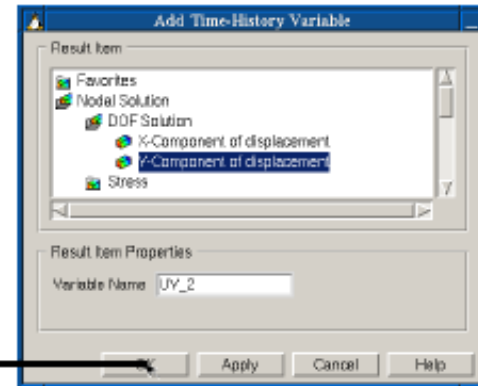
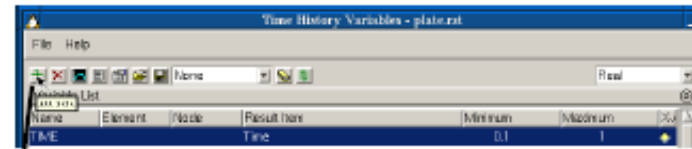
- In the steady state we will only use the General Postproc (/POST1)
- We can view results using commands, but it is often easier to do this graphically
General Postproc > Read results > Last Set
- A few examples:
General Postproc > Plot results > Contour plot > Nodal Solution



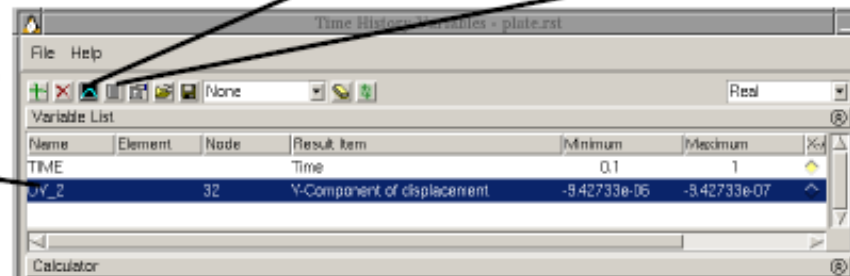
- We can do this through:
List > Results > Nodal solution
General Postproc > List results > Nodal solution
- This will list all selected nodes, so we might want to select fewer nodes graphically or with **NSEL**

Пост-процессорная обработка результатов в Time history postprocessor для нестационарного анализа (transient analysis)

- TimeHist Postproc has a graphical interface which lets us add variables and plot them
- If we want to look at the variation of displacement of a node with the applied load we can add it as a variable
- Type **eplo** to plot the elements of your model, make sure you can see the region you're interested in clearly
- First click on **+** to add a new variable
- Then select the results type to add and click OK
- Then select a node from the model and click OK
- You should see a new variable has appeared



PLOT variable LIST variable



Графическое отображение процесса решения при нестационарном анализе

